ePLAN®5

Training Manual
Basic Training – Version 5.50

August 2003

Note: The hardware requirements indicated by the operating system manufacturers must be considered. Hardware combinations that according to the manufacturer's specification cause malfunctions may also have an effect on the operation of EPLAN LOGOCAD TRIGA. Therefore an accurate basic installation of hardware, operating system and hardware drivers (e. g. graphics board driver etc.) is essential for a smooth operation of EPLAN / LOGOCAD TRIGA. Therefore the main conditions and areas of application specified by the hardware manufacturers also apply to the operation of EPLAN / LOGOCAD TRIGA.

EPLAN 5 Basic Training

Topics and Contents

Basic Working Methods

- Defining personal directory settings
- Editing projects
- EPLAN page types
- Using the working window of graphics editing
- Using macros

Automatic Evaluations

- Generating cross-references and associated lists
- Contactor selection
- Terminals
- Cables
- Creating a table of contents
- Backing up data

Exercises

- Creating a sample project incl. data backup

EPLAN 5 Basic Training

Defining Personal Directory Settings

The **Directories** option under the personal parameters is used to specify the directories in which the programs should search for specific files. The individual EPLAN files are distributed among the following different directories:

Projects:
In EPLAN, projects are saved in the directory `\EPLAN4\P`. This directory is created during the installation and cannot be changed by the user. But it is possible to create subdirectories under this directory in order to make project management easier.

Master data:
These include symbol files, plot frames, parameters, and plot forms. Print forms are filed in a separate directory (refer to "Print, control forms").

Macros:
Macros must be filed in a directory below `EPLAN4\M`, otherwise the program will not find them.

Parts data:
These are the files for the system-internal stock management. These include master parts files.

Print, control forms:
These are terminal, cable, potential forms, and contactor lists as well as the forms for the system-specific parts management program Plot forms are located in the master data directory!

In addition, this dialog box displays the directories for:

- **Personal and workstation data**
- **Temporary data**.

These two settings serve for checking purposes and cannot be changed. In all fields you can right-click to launch the pop-up menu and to assign the standard entry to the respective field.

ePLAN ® 5

EPLAN 5 Basic Training
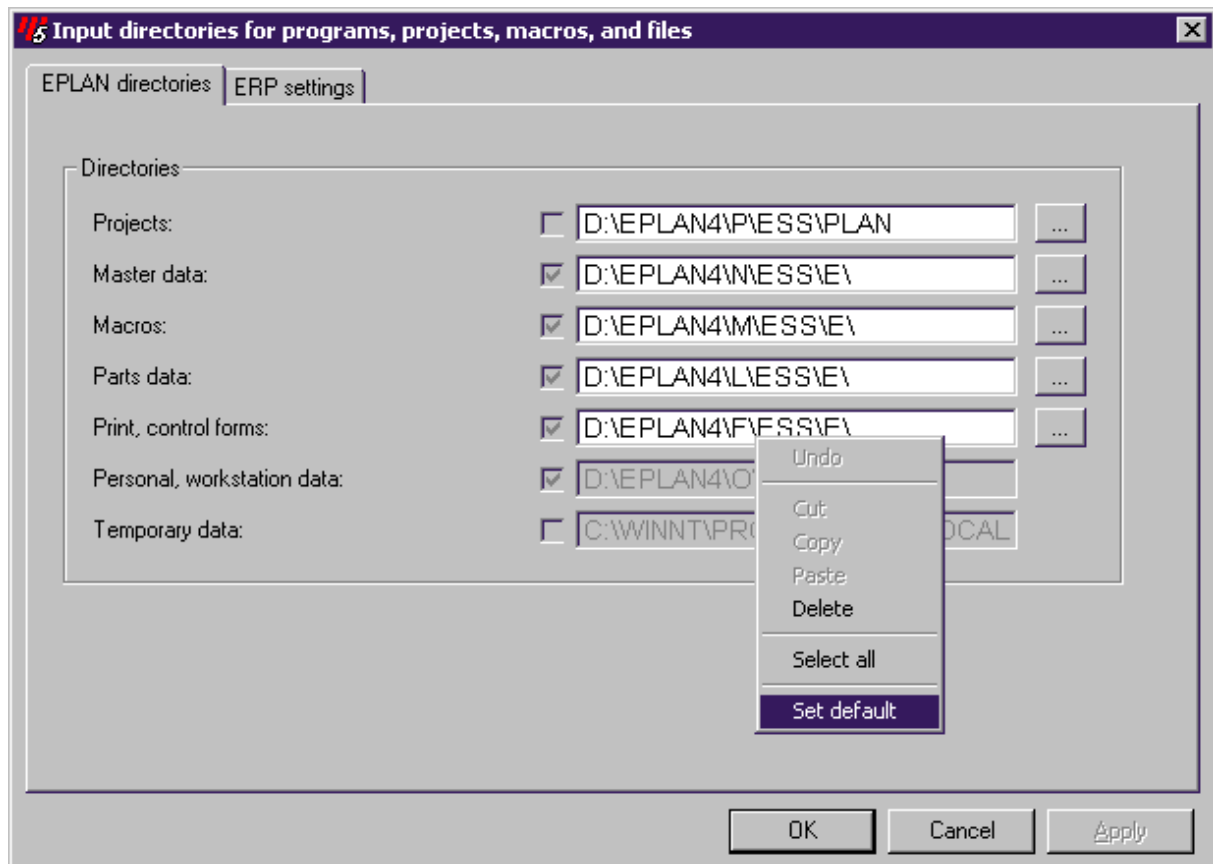
Use the **[...]** command button in order to branch to the Directory selection dialog box and to select the directory interactively.

If you enter a directory that does not yet exist manually, the program will prompt you if you want to create it now. The check boxes before the directory entries indicate whether the respective directory exists or not.

☞ When specifying the directories ensure that they are positioned within the `EPLAN4` directory structure. All the projects must, for example, exist below the directory `\EPLAN4\P\` or its subdirectory. If you specify an invalid directory, EPLAN draws your attention to this fact when you leave the dialog box. If you have specified several directories which do not exist, you can decide whether EPLAN should create all of them or only individual ones.

EPLAN 5 Basic Training

Editing Projects

The following overview provides a list of functions that are available in the Project selection dialog box. For the most part, these functions can be carried out via corresponding buttons in the dialog box as well as via menu options in a pop-up menu. To call up the pop-up menu, the cursor must be located in the directory tree of the Project selection dialog box.
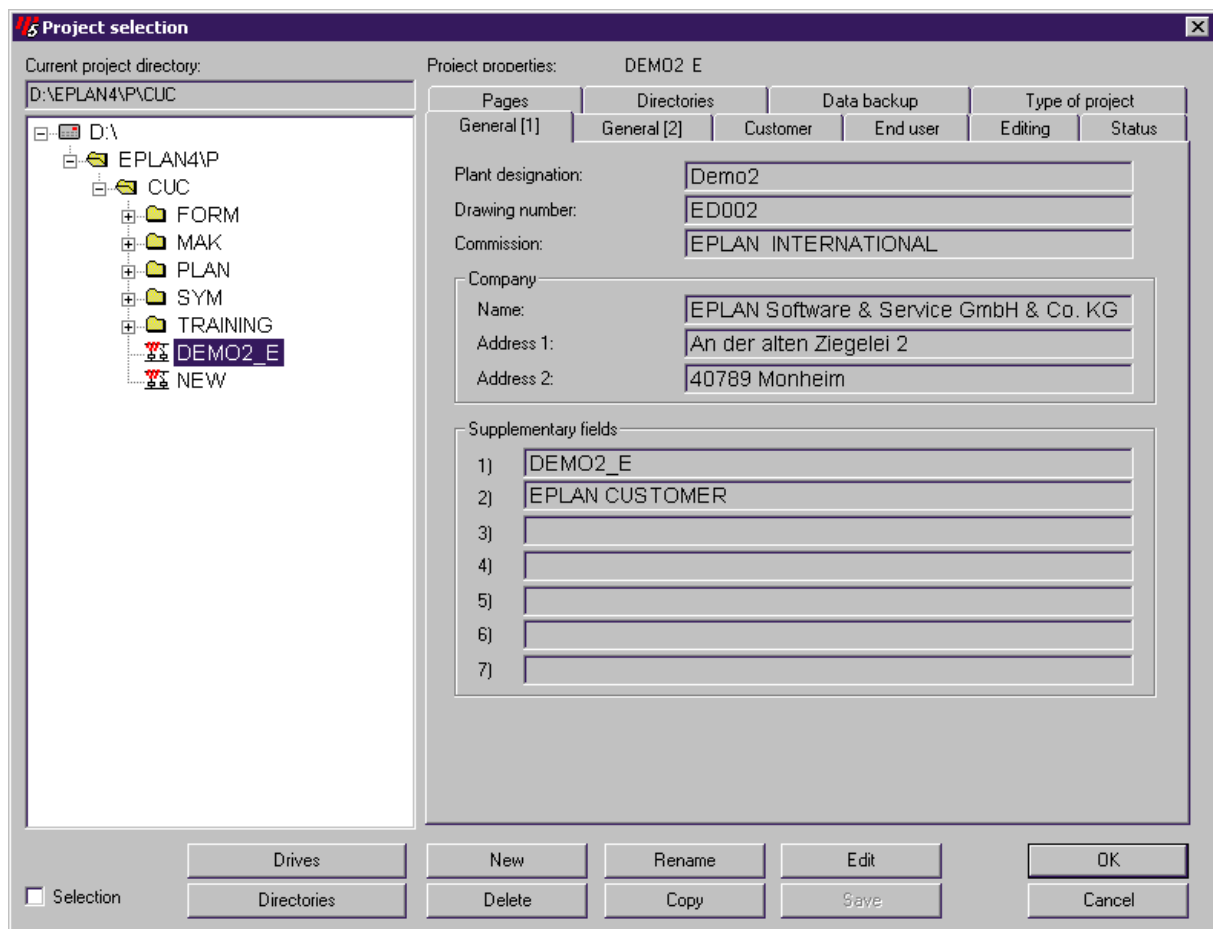
👉 Please note that the availability of the functions depends on whether you have selected a directory or a `project` and if you are currently editing the project properties.

EPLAN 5 Basic Training

Function	Action
New	Create a project/directory This option can always be activated. Depending on the subsequent option you select, either a new project is created in the corresponding (sub)directory or a new subdirectory is created.
Rename	Rename projects/directories Using this function you can rename the project or directory currently highlighted in the directory tree.
Copy	Copy projects Activate this option and enter the drive, path, and name of the project to be created in the subsequent dialog box.
Delete	Delete projects/directories Use this functions to delete projects or empty directories. A prompt for confirmation is displayed when you select this option.
Edit	Edit projects This function is used to edit the project properties of the currently selected project.
Save	Save projects Use this function to save the project properties you entered. The corresponding button or menu option is activated when a new project is created, when projects are copied, or when project properties are edited.
Drives	Change the drive for the project selection Use this function to have all drives displayed in the directory tree that contain EPLAN projects. You can then change the drive for the project selection.

EPLAN 5 Basic Training

Function	Action
Directories	Change directories Use this function to influence the access to macros, master and parts data as well as the print and controls form for the currently opened project. This option is only available as a button.
Selection	Select projects / delete selection criteria Select the **Create** submenu option from the **Selection** menu item if you want to preselect projects for the directory tree. Select the **Selection > Delete** menu items to remove the specified selection criteria again.
Compress	Compress projects This function is used to compress the currently selected project.
Send by e-mail	Send projects by e-mail Select this option from the pop-up menu in order to send the selected project by e-mail.
Show user	If you select this function, you will get the information which users are currently editing the project selected in the project management.

👉 Please note that the Show user function is part of an add-on module. Only when the license for the module has been purchased can you carry out this function.

ePLAN® 5

EPLAN 5 Basic Training

Projects and Directories

EPLAN files projects in the `\EPLAN4\P` directory which is created when you install the program. This directory cannot be altered. You can, however, create subdirectories under this directory in order to make project management easier. These subdirectories will be referred to as "project directories".

Projects include shared data not stored in the project directory. This is information used by all projects and includes macros, master and parts data, as well as print forms.
During the installation, a directory with your customer code is created in the various directories. In order to keep your data separate from other customer-specific data, you should create additional directories for each customer.

> 💡 If you use special macros, parts data, or print and control forms for customer "Smith", you should create the following directories for this information:

`\EPLAN4\N\SMITH`	For master data.
`\EPLAN4\M\SMITH`	For macros
`\EPLAN4\L\SMITH`	For parts data
`\EPLAN4\F\SMITH`	For print and control forms.

Specifying the Project Type

When you create a project you also specify the purpose to which the project is to be used in the **project type**. The following project types are available:

- **Schematic project**
- **Symbol project:** for editing symbols
- **Form project:** for editing plot forms (for terminal diagrams, interconnect diagrams, bills of materials etc.)
- **Macro library project:** for editing macros which are to be used in the Schematics Generator.
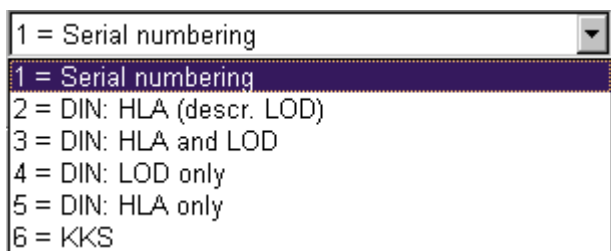
ePLAN®5

EPLAN 5 Basic Training

In EPLAN you will most often use schematic pages. However, a project can generally be any of the above-mentioned project types. The project type can also be changed subsequently: Thus you can change a schematic project at any time into a symbol or form project.

This means that you can include, for example, sheets for plot forms, terminal diagrams, interconnect diagrams, etc. in a schematic project and use them correspondingly when required.

Specifying the Page Numbering Type

The page numbering type you specify determines how the project pages will be managed. EPLAN supports the following page numbering types which you can specify via the dropdown list box in the Page numbering type field:

```
1 = Serial numbering                        ▼
1 = Serial numbering
2 = DIN: HLA (descr. LOD)
3 = DIN: HLA and LOD
4 = DIN: LOD only
5 = DIN: HLA only
6 = KKS
```

- Serial numbering
- DIN page numbering
- KKS page numbering (German power station numbering system).

The individual page numbering types are explained in the following sections.

⚠ Once you have created a project, you can no longer change the page numbering type, because the structure of the database in which the project is stored depends on the selected page numbering type. The field is therefore grayed and cannot be edited when you call up the "Project type" tab again later.

EPLAN 5 Basic Training

Serial Page Numbering

In this numbering type the pages are numbered consecutively. The page numbers must lie within the range of 1 to 99,999.

In addition to the page number you can specify a subpage consisting of exactly one number or character. This is advisable in particular if you have to include **additional** pages when modifying an existing diagram or a list.

> Examples are:
> ```
> 10
> 10.3
> 24.C
> ```

DIN Page Numbering

In the case of DIN projects the page designation can include a higher-level assignment, a location designation or both, depending on the numbering type selected.

EPLAN provides the following numbering types for the resultant combination possibilities:

- Higher-level assignment and describing location designation
- Higher-level assignment and location designation
- Location designation only
- Higher-level assignment only.

The two upper types differ only in as far as a local designation specified at the type "Higher-level assignment and describing location designation" is **not** used to identify a page. Only the higher-level assignment is used.

In the second type "Higher-level assignment and location designation" **both** identifiers are used for identification. In the other types the respective identifier used exclusively - "Location designation only" or "Higher-level assignment only" - serves to identify a page.

EPLAN 5 Basic Training

Sequence of the Identifiers

As a rule the individual identifiers are to be used in the following order for a page designation to DIN:

- Higher-level assignment (if appropriate, subpage)
- Location designation (if appropriate, subpage)
- Page number (if appropriate, subpage)

At the type "Higher-level assignment and describing location designation" at least one of the two identifiers (higher-level assignment, page number) must identify the page uniquely. The page number is unique in the following example:

```
=HLA.1+LOD.1/1
=HLA.1+LOD.1/2
```

At the type "Higher-level assignment and location designation" at least one of the three identifiers (higher-level assignment, location designation, page number) must identify the page uniquely. The higher-level assignment is unique in the following example:

```
=HLA.1+LOD.1/1
=HLA.2+LOD.1/1
```

At the type "Only location designation" at least one of the two identifiers (location designation, page number) must identify the page uniquely. The location designation is unique in the following example:

```
+LOD.1/1
+LOD.2/1
```

At the type "Higher-level assignment only" at least one of the two identifiers (higher-level assignment, page number) must identify the page uniquely. The page number is unique in the following example:

```
=HLA.1/1
=HLA.1/2
```

EPLAN 5 Basic Training

Page Numbering according to KKS

Page numbering to the German KKS for numbering power stations is carried out using the higher-level assignment and location designation. These identifiers are **preceded** by one identifier each for the document type and the documentation type.

A page designation to KKS thus consists of:

- Document type
- Documentation type
- Higher-level assignment (if appropriate subpage)
- Location designation (if appropriate subpage)
- Page number (if appropriate subpage)

Ensure that the page designations differ by **at least one** identifier.

> The following expressions are thus valid page designations to KKS:
> ```
> YD=HLA.1+LOD.1/1
> YE=HLA.1+LOD.4/1
> ```

Specifying Subpages

In all page numbering types you can assign so-called **subpages** to the existing project pages. This can be relevant, for example, if you have to insert additional pages into an existing project.
Subpages are created by specifying the number of the corresponding main page, followed by a numeral or a character.

In a project with serial numbering 10.3 or 24.C would therefore be valid designations for subpages.
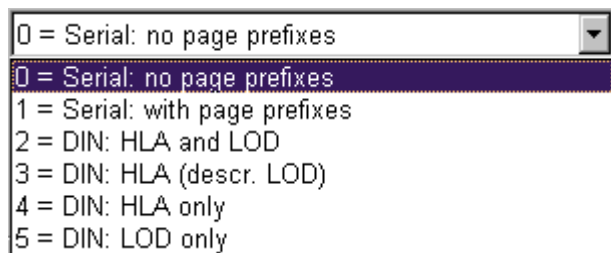
EPLAN 5 Basic Training

Specifying Designation Methods

Designations or identifiers are used to identify devices within an electrical project. Devices include individual components, equipment, assemblies, functional units, etc. that are each represented in the schematic by a graphics symbol. The identifier provides the following information:

- Interrelationships between the device and other parts of the plant
- Mounting location of the device
- Identification of the device by its type, counter number and function.

EPLAN supports the designation methods for devices, terminals, cables, and interruption points specified in DIN 40719, Part 2.

EPLAN supports six device designations methods.

```
0 = Serial: no page prefixes          ▼
0 = Serial: no page prefixes
1 = Serial: with page prefixes
2 = DIN: HLA and LOD
3 = DIN: HLA (descr. LOD)
4 = DIN: HLA only
5 = DIN: LOD only
```

EPLAN 5 Basic Training

In the case of **serial numbering** the devices are identified exclusively by the device designation. If you use the variation "with page prefix", the number of the page which contains the device is prefixed to each device during evaluation runs.

> 💡 For example, F1 becomes 19F1 if this device is used on Page 19 .

The same variations already known to you from page numbering are available for **numbering to DIN**. Depending on the selected numbering method the higher-level assignments and location designations are positioned at the front.

> 💡 A component K1 which is to be displayed with the selected numbering method "With identifying higher-level assignment and location designation" is, for example, displayed as =HLA.1+LOD.1-K1.

> 👉 Note that the specified numbering methods only have an effect on automatic evaluation runs. Only the device designation is displayed in the schematic.

Specifying the Designation Methods for Terminals, Cables, and Interruption Points

EPLAN supports seven different designation methods for terminals, cables and interruption points.

EPLAN 5 Basic Training

EPLAN Page Types

As a rule an EPLAN project consists of pages of different **types**. The page types can be differentiated according to several criteria:

- Logic page or graphics page?
- Page created interactively or automatically?

Logic pages:
If a page was created as a **logic page**, this means that EPLAN can process the information saved on it for the automatic generation of lists and diagrams.

Graphics pages:
In general, logic information cannot be stored on graphics pages for the automatic evaluation.

Interactive pages:
These are pages that you create and edit yourself, such as schematic pages or free graphics pages.

Automatic pages:
These pages are automatically created and processed by EPLAN. This includes all pages whose data are first created during generation runs and then output as project pages, e.g. terminal and interconnect diagrams, etc.

EPLAN 5 Basic Training

On the basis of the criteria mentioned above, the various page types can be assigned to the following groups:

Group	Page types
Interactive logic pages	Schematic, PLC page, Single-pole display
Interactive graphics page	Free graphics, Control-panel layout, Plot frame creation, Title page/cover sheet, PLC overview
Automatic graphics pages	Table of contents, Terminal diagram, Terminal line-up diagram, Terminal-connection diagram, Interconnect diagram, Bill of materials, Device list, Terminal-strip overview, Cable overview, Purchase-order list, Wiring and wire list
Special pages	External document

All pages belonging to one specific group are indicated by one standard symbol in the page overview. For example, all automatic graphics pages have the symbol (see the section "Symbols used in the page overview"). An exception to this rule is the interactive logic pages, as a symbol is used for pages of the Schematic and PLC page type and another symbol for pages of the Single-pole display page type.

The following section provides an overview of the various EPLAN page types.

EPLAN 5 Basic Training

Page Type Overview

You determine the page type of a page while creating it. We recommend **not** to change the page type when processing of the page properties later, since information may get lost while doing so.

Whether menu items and functions can be activated in graphics editing depends on the page type. For example, the insertion of symbols from symbol files is only possible on logic pages (schematic pages, PLC pages, and pages in single-pole display).

👉 The **J, K, L, M, N, O, P, R, S, U, V** page types (automatic graphics pages) are automatically created by EPLAN. Only if you want to edit the forms in a forms project may it be necessary to manually create pages of this type.

The following page types are available in EPLAN:

A = Schematic:
Schematic pages can access special functions associated with the creation and evaluation of schematics. This includes inserting symbols with automatic wire as well as functions taken into account during generation runs for terminal and interconnect diagrams or for cross-referencing.

EPLAN 5 Basic Training

Using the Working Window of Graphics Editing

The graphics editing module provides a specific window type - the working window - for drawing schematics. The working window (also called "graphics screen" in the documentation) are default Windows windows offering the familiar functions such as Move, Minimize etc. In contrast to the two other window types (page overview, message window) in graphics editing, it can only be placed within the main window.

A working window shows the graphics contents of an open page. If you have modified an open page, EPLAN displays an asterisk in the title bar of the working window to indicate this.

🔆 *Example of the title bar of a modified page*

`[EPL5001 Graphics editing =AP+ST1/13  *]`

EPLAN allows you to view **several pages** at the same time on the screen. For this purpose you can open several working windows (see the section on "Opening pages in a new window" in this User Guide). Even if several working windows are open at the same time, pages can be edited in only one window at a time. The window containing the page in which the current cursor movements and functions are carried out is called the "active window". You can recognize this window by its colored title bar.

EPLAN 5 Basic Training

Switching between Several Open Working Windows

If you want to change from one working window to another open working window, simply click on it. In addition, the [Ctrl] [⇆] key combination can be used for this purpose.
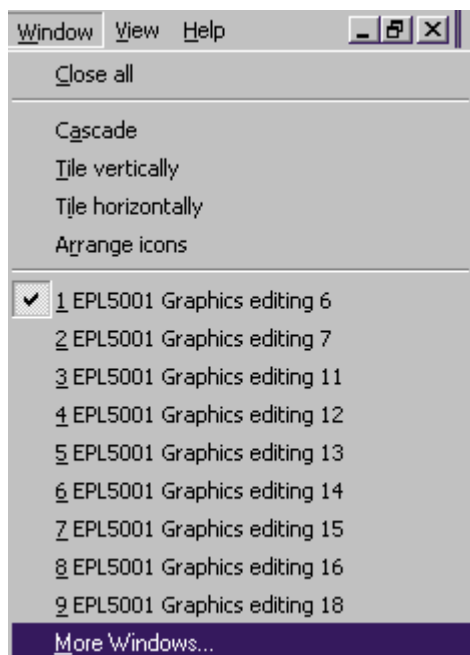
Project > Graphics editing, output > Window

It is also possible to select the desired window via the **Window** menu. All open working windows are listed in this menu. The active, topmost window is marked by a preceding checkmark (✔). In this list, a modified page is marked with an asterisk (see the highlighted third menu entry in the example). If more than 9 working windows or pages are open, you can select the **More windows...** option to call up a dialog box in which all open pages are available.

💡 Example for the **Window** menu with several open pages

Window View Help
Close all
Cascade
Tile vertically
Tile horizontally
Arrange icons
✔ 1 EPL5001 Graphics editing 6
2 EPL5001 Graphics editing 7
3 EPL5001 Graphics editing 11
4 EPL5001 Graphics editing 12
5 EPL5001 Graphics editing 13
6 EPL5001 Graphics editing 14
7 EPL5001 Graphics editing 15
8 EPL5001 Graphics editing 16
9 EPL5001 Graphics editing 18
More Windows...

ePLAN® 5

Closing all Working Windows

Project > Graphics editing, output > Window > Close all

To close all open pages or working windows at one go, select the **Close all** option from the **Window** menu.

EPLAN 5 Basic Training

1. Editing Schematics

The ability to graphically display devices as symbols is the most important function in the creation of schematics. In EPLAN you can only graphically show devices as symbols on "logic" type pages (for example, on the Schematic or Single-pole page type). In the following sections we will explain the multipole display. For information on schematics in single-pole display, please refer to the section entitled "Creating Schematics in Single-pole display".

💡 Schematic extract with examples for the display of devices
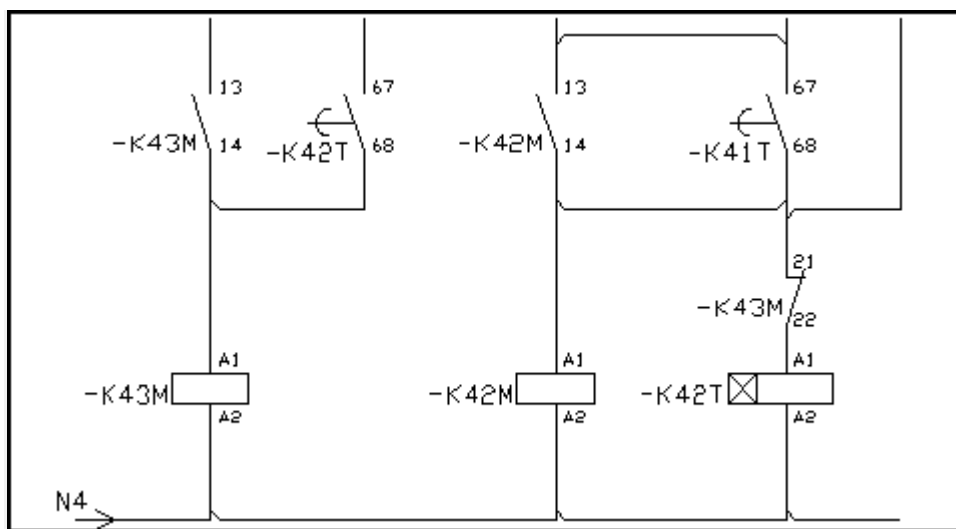
Terminology:
The following terms are particularly important when working with symbols and devices:

Symbols, device designations:
Electrical and mechanical devices are represented by graphics **symbols** in EPLAN. A device in the project is identified by EPLAN by means of the **device designation** (DD) you always have to specify in order to receive error-free evaluations.

Symbol files
EPLAN manages most symbols in symbol libraries - the so-called **symbol files**. You can extend or edit them. Apart from the symbols you can insert from symbol files into the schematic, there are symbols that are not linked to a symbol file, for example the black box, the interruption point, or the device-end terminal.

Symbol properties
When inserting a symbol into the schematic, EPLAN opens a dialog box into which

ePLAN ®**5**

you must enter symbol-specific texts or identifiers. These data are called **symbol properties**. EPLAN evaluates the symbol properties during the various generation runs. One of the most important symbol properties is the device designation. Depending on the symbol you insert into the schematic, EPLAN displays various fields for inserting symbol properties.

Wires

If the connecting points for device symbols lie opposite one another either vertically or horizontally, connections between the symbols are generated automatically ("autoconnecting"). Angles and nodes are used to branch and divide wires. You can use interruption points to continue a wire over a number of pages of the schematic.

Texts and hyperlinks

In addition to devices and wirings, it is also possible to insert special objects such as texts and hyperlinks into the schematic in order to get a documentation that is structured more clearly and to obtain additional information.

A hyperlink is a text inserted in the schematic providing a link to a document.

Using Macros

Macros are used in EPLAN to facilitate your work, as is the case in other programs. You can combine several steps, such as the insertion of various components and schematic elements with the assignment of device designations, to one step, that is save as a macro and insert at a later point.

In general, macros consist of any sections of schematic or graphics pages in EPLAN or include the entire contents of a whole project page. Furthermore, there are various other macros, which provide specific functions.
The following sections first introduce these various macro types.

Then, further sections will follow dealing with the following subjects:

- Defining a new macro
- Using macros

Finally you will be familiarized with a specialized application of macros - the Schematics Generator. With the Schematics Generator you can use standardized macros to create your plant documentation without any drawing effort.

Macro types:
In EPLAN you can use six basic macro types and three special macro types. The basic macro types are:

Macro type	Character	Schematic	Graphics
Page macro	S	x	x
Window macro	F	x	x
PLC macro	L	x	-
Path macro	P	x	-
Variable macro	V	-	x
Combined macro	Z	-	x

ePLAN® 5

EPLAN 5 Basic Training

Every macro is saved **in a separate file** in the directory which you set for saving macros in the parameter management module. The extension of a macro file name provides information on the type and possible uses of the macro:

- The first character of the extension is always `M` (for "macro").
- The second character specifies the macro type - this is the Character column in the table.
- The third character of the extension specifies the page type on which the macro is to be inserted: `E` for schematics pages and `F` for graphics pages. The table columns Schematic and Graphics indicate whether a certain macro type is suitable for the respective page type.

Not every type of macro can be created and inserted **on every type of page**. In addition to the specified types there are further special macro types for which the above rule on the file extension does not apply:

Macro type	Extension	Schematic	Graphics
Plot frame	SK?	x	x
Symbol macro	MYS	x	-
Control panel macro	MRS	-	x
Rittal control panel macro	DXF		x

Symbol Macros:

A symbol macro combines several individual symbols which represent a unit into **one** object. Symbol macros are intended as a supplement to the symbol files. They are saved with the `.mys` file extension.

Utilities > Parameters > Personal > Graphics

If you use the personal parameter "Symbol macros directory", you can specify that symbol macros are **not** to be filed below the directory `N` for master data, but rather, below the actual macro directory `M`.

For saving symbol macros we recommend to file them in a directory that is named after the used symbol file in order to have a good overview.

`\EPLAN4\N\<customer code>\DIC_WUP<X>\M3.MYS`
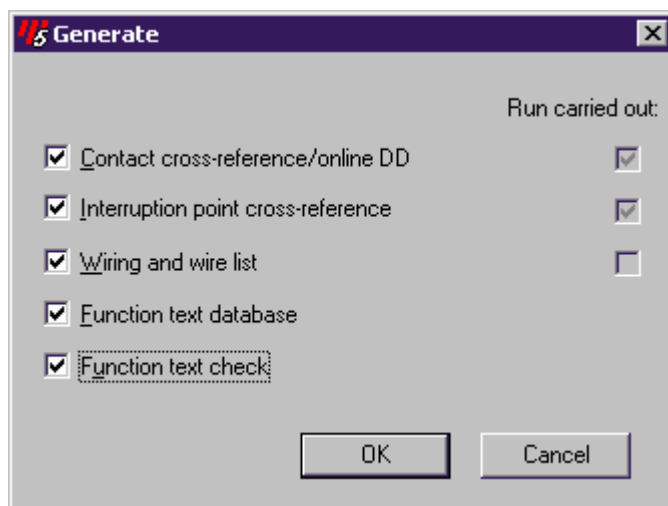
ePLAN®5

EPLAN 5 Basic Training

Generating Cross-references and Associated Lists

Generate > Cross-references/wiring list

In this chapter you will learn how to update cross-reference data with various generation runs and how to carry out several other functions. The respective module is accessed by selecting the menu path **Generate > Cross-references/wiring list** from the main menu.

Select **Generate > Generate** to open a selection dialog box. Here, you can decide whether you want to start the listed generation functions as shown in the following figure, individually or together.

If a generation run has been carried out before, EPLAN indicates this by setting a check in the grayed fields on the right of the dialog box.

ePLAN® 5

EPLAN 5 Basic Training

In the following sections you will find detailed information on the topics:

- Generating cross-references
- Creating wiring lists
- Updating the text database and checking function texts
- Creating and deleting wire numberings
- Automatically correcting cross-reference errors in the schematic
- Creating a list of special components
- Deleting cross-reference databases
- Editing cross-reference data
- Outputting formatted cross-references
- Deleting the contactor contact arrangement in the schematic
- Outputting wiring lists as graphics pages
- Evaluating messages produced during cross-reference generation
- Setting parameters for the cross-reference display and output
- Online cross-references when copying schematic pages
- Online-updating of cross-references when renaming and deleting schematic pages.

Generating cross-references:
EPLAN produces two types of cross-references in the schematic:

- **Contact cross-references**
- **Interruption point cross-references**.

EPLAN 5 Basic Training

The program automatically (online) inserts both cross-references into the schematic while the project is being edited. Nevertheless we advise you to update the contact and interruption point cross-references with a generation run. This is of particular importance if:

- Cross-reference changes result from loading macros and the individual symbols/interruption points have not been confirmed with [⏎] in the graphics editing module
- You have modified page numbers, path numbers, or positions via the numbering program
- Projects have been created or modified by other programs.

In addition, EPLAN creates an exact error list when generating cross-references. Error-free cross-references are a **prerequisite** for evaluating the following program sections:

- Wiring list
- Contactor selection
- Terminal and interconnect diagram
- Device list.

You can use the correction function to have some cross-reference errors (as a general rule, those errors that mostly occur) automatically eliminated.
Please refer to the following sections for more information on the topics:

- Generating Contact Cross-references
- Creating Interruption Point Cross-references
- Generating Cross-references according to IEC 1082-1.

Generating Contact Cross-references

This subchapter is about the creation of contact cross-references. First, you will get to know the various types of contact cross-references, and afterwards, you will learn how to generate contact cross-references according to the IEC 1082-1 standard. Finally, we will describe how to start a contact cross-reference run.
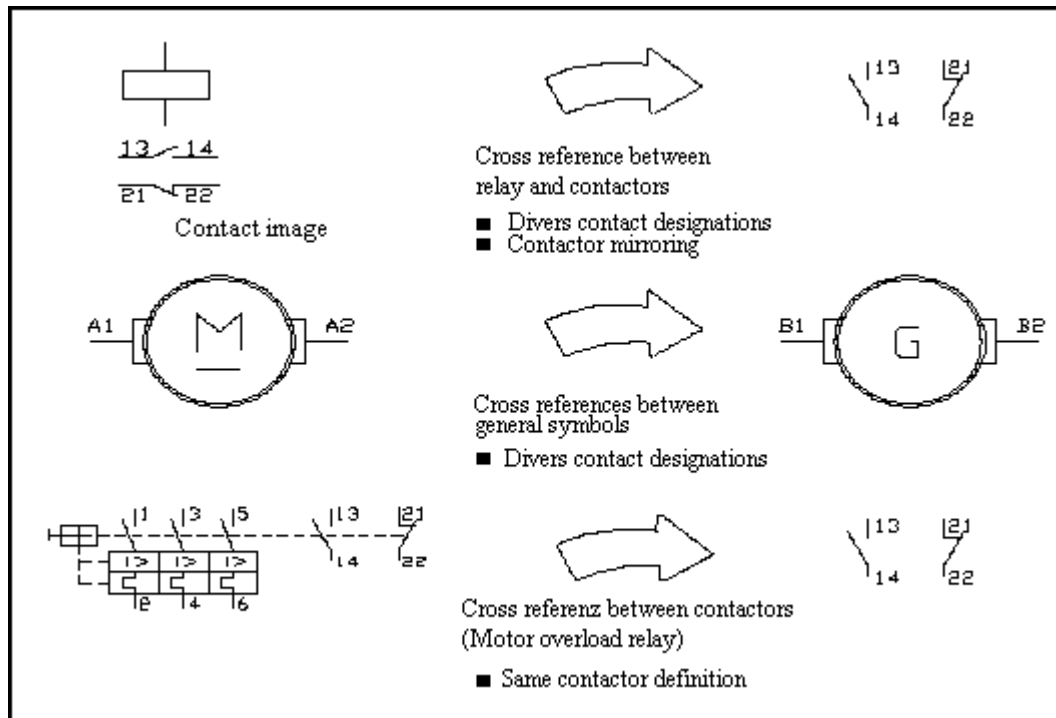
Contact Cross-reference Types

It is often necessary to display the same device several times. The fact that the corresponding symbols belong to this device is recognized by the fact that all the symbols have the same device designation and are represented by the corresponding symbol type.

The contact cross-reference shows where the other part of the device can be found in the schematic.

There are the following contact cross-reference types:

- **Contactor cross-references:** (cross-references between the contactor coils and the contacts)
- **Symbol cross-references:** (cross-references between general symbols)
- **Pair cross-references:** (cross-references between contacts).
- **Counter cross-references:** (cross-references between symbols that were made able to be cross-referenced)
- **Contact image:** (image of all the contacts that are cross-referenced with a contactor coil).

The individual cross-references are generated in different ways:

EPLAN 5 Basic Training
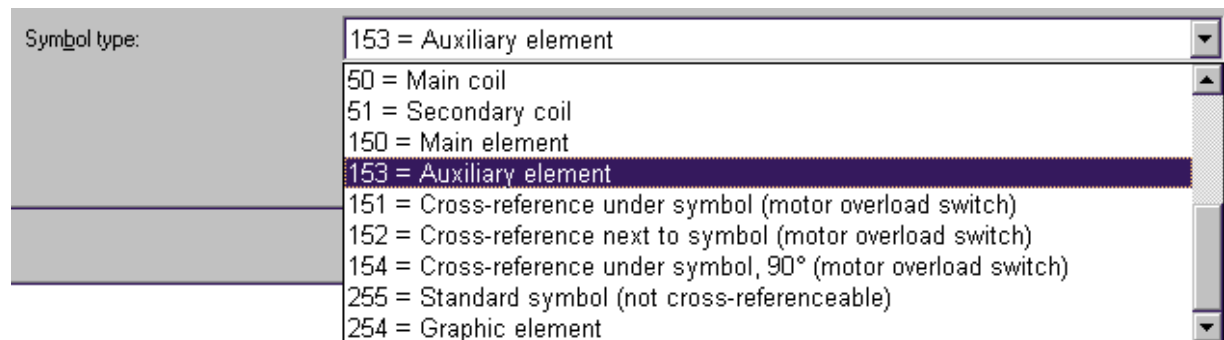
Contactor Cross-references

Coils and contacts are marked correspondingly when the symbol is designed in the symbol editor. Here, the symbol type is specified which defines the symbol as coil or contact symbol. A cross-reference to a coil symbol is drawn in the form of a contact image whose appearance is defined in the graphics parameters.

Thus, coils and contacts are "by their nature" able to be cross-referenced, while other cross-references are generated as a result of the respective elements having been "made able to be cross-referenced".

Symbol Cross-references

General symbols can have a cross-reference assigned during schematic editing (can be made able to be cross-referenced). This is the only case in which two symbols (graphics symbols) can have the same device designation. The connections on the symbols representing the device connections must have different connection designations.

In symbol cross-referencing the individual symbols are identified as having a main and one or more auxiliary elements. The settings required for this can be specified in the Symbol properties input window that you call up in the Symbol type dropdown list. Depending on the symbol type this list may contain different elements.

Symbol type:	153 = Auxiliary element ▼
	50 = Main coil
	51 = Secondary coil
	150 = Main element
	153 = Auxiliary element
	151 = Cross-reference under symbol (motor overload switch)
	152 = Cross-reference next to symbol (motor overload switch)
	154 = Cross-reference under symbol, 90° (motor overload switch)
	255 = Standard symbol (not cross-referenceable)
	254 = Graphic element

EPLAN 5 Basic Training

The following table lists the possible identifiers for the symbol type of the main and auxiliary elements:

Symbol type	Identifier
Standard symbol (not cross-referenceable)	255
Symbol as main element cross-referenceable	150
Symbol as auxiliary element cross-referenceable	153

You can only enter part-specific (component-specific) data, such as the part number, for a main element. You can assign n+1 components with the same DD to act as auxiliary elements.

Pair Cross-references at Contacts

Cross-references between contacts are used to represent the interdependencies between contacts when the component in question is not a contactor. This applies for instance to the auxiliary contacts of a motor overload switch. In this case, two contacts with the same connection designations are used in the schematic. That is, the same DD or the same contact occurs twice.

Pair cross-references are entered in the same way as symbol cross-references:

Symbol type	Identifier
Contact as a contactor contact	0-15
Contact with cross-reference below the symbol (main element)	151
Contact with cross-reference next to the symbol	152
Contact with cross-reference below the symbol, but rotated by 90°	154

☞ Notes:
Note that two symbols with both the same device designation and the same connection designation may only be used if contacts have been pair cross-referenced. In addition, the use of the same device designation means that only half as many pairs can be used as the maximum number specified in the "Maximum number of contactor contacts" parameter.
Furthermore please note that **it is not allowed** to use symbol and pair cross-references at the same time for one device designation.

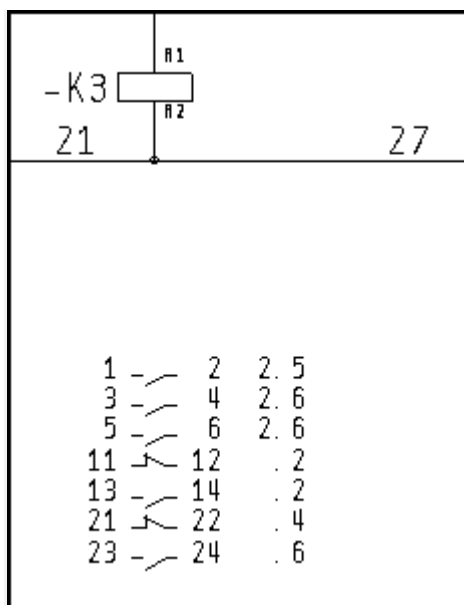# EPLAN 5 Basic Training

Counter Cross-reference

The cross-references of contacts are placed below the device designation. For symbols that have been made cross-referenceable, the same method is used as for contacts.

Contact Image

A contact image containing all cross-references to a coil is drawn below the respective coil in the schematic. The appearance of the contact image is governed by parameter settings. The contacts are listed in ascending alphanumeric sequence of the connection designations. An exception to this are changeover contacts.

💡 Extract from a schematic page with contact image

Starting a Contact Cross-reference Run

Proceed as follows to start the generation process: Select **Generate > Generate** in the Cross-references module. Then activate the Contact cross-reference / online DD check box in the subsequent Generate dialog box. A checkmark in the grayed Run carried out field indicates that a contact cross-reference run has been carried out before. Once you confirm the dialog box by clicking **[OK]**, EPLAN starts the contact cross-reference run. If contact cross-reference errors occur in the project, EPLAN outputs messages during the evaluation.

Creating Interruption Point Cross-references

This subchapter is about the creation of interruption point cross-references. First of all, the terminology of the interruption point cross-reference is explained in the following section. Finally, we will describe how to start a interruption point cross-reference run.

Terminology of the Interruption Point Cross-reference

Interruption points are used to represent wires that go across more than one page. The cross-references represent the reference to the page in the schematic on which the wire continues. Target tracking is carried out at the same time. Target tracking means that a target is searched for each interruption point. The target leads from this interruption point to the next component.

EPLAN 5 Basic Training

The following section defines some of the terms used in the creation of interruption point cross-references:

Target:
Targets are defined as all those symbols that represent a device. Other graphics symbols such as angles and nodes cannot be targets, although they influence target tracking.

Target tracking:
EPLAN searches for a target for every interruption point. The search continues along the generated wires until a target is detected.
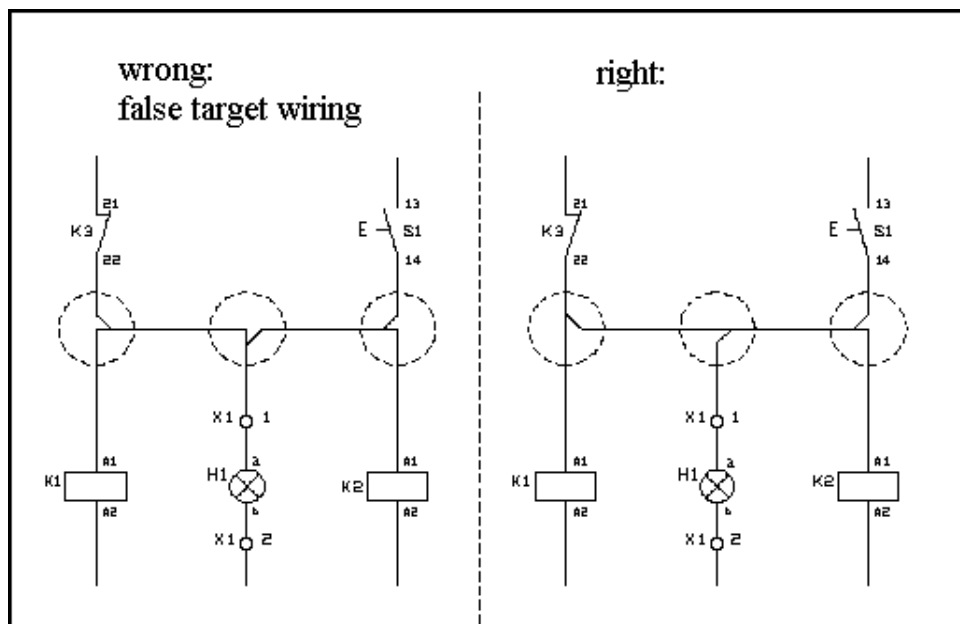
Counter target:
Where two interruption points that belong together appear on different pages, the target DD always has a corresponding counter-target.

Target wiring:
The target information for the wiring is important for target tracking. This is particularly true when nodes are used. If point wiring was used to draw nodes in the graphics, a sensible definition for target tracking must be included so that the wiring will produce the desired results.

💡 Incorrect / Correct target wiring

Target Tracking along Interruption Points

Target tracking along interruption points is important for the following functions:

- **Display of the counter-target**: You can use parameters to change the information at the interruption point. Among other things you can also have the counter-target displayed.
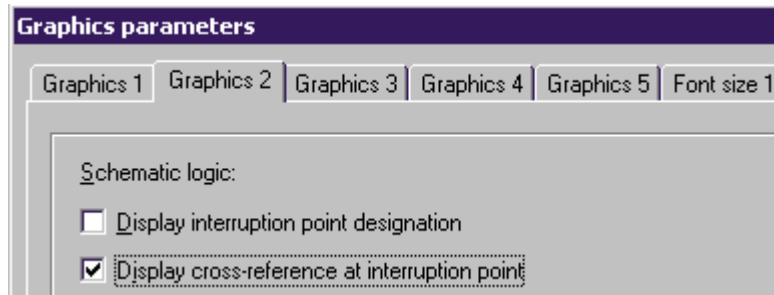
- **Determination of the wiring target**: If a wire between interruption points leads to another schematic page, the relationship between targets must be determined in order for the wiring list to be generated.
- **Determination of the terminal targets**: If an interruption point is connected to a terminal, the interruption point's counter-target will be entered as the target during the evaluation of the terminal data.

Identifier for the Interruption Point Check

Use the "Identifier for interruption point check" parameter to specify what kind of interruption point check EPLAN is to carry out during the cross-reference run. The individual options are explained in detail in the chapter entitled "Setting Parameters for the Cross-references".

EPLAN 5 Basic Training

Starting an Interruption Point Cross-reference Run

Interruption point cross-references are generated in a procedure analogous to that for contact cross-references. Simply select **Generate > Generate** in the cross-reference module, and activate the Interruption point cross-reference check box in the subsequent dialog box. Here, too, a checkmark in the Run carried out field indicates that you have already generated interruption point cross-references at an earlier point in time. Once you confirm the dialog box by clicking **[OK]**, EPLAN starts the interruption point cross-reference run. If interruption point cross-reference errors occur in the project, EPLAN outputs messages during the evaluation.

Generating Cross-references according to IEC 1082-1

Up to now, a cross-reference has in general been represented by a page and a path specification, separated by a specific character.
The so-called path specification is the subdivision of a schematic page into several columns marked by a number or a letter.

In many cases, an additional subdivision in line areas is required. This requirement is based on the **IEC 1082-1** standard, which corresponds to the EN 61082-1 standard and which is supported by EPLAN as from version 5.50.

EPLAN now adds one letter to the cross-reference, which is placed **ahead** of the path specification. This letter specifies the line area of the counter element.
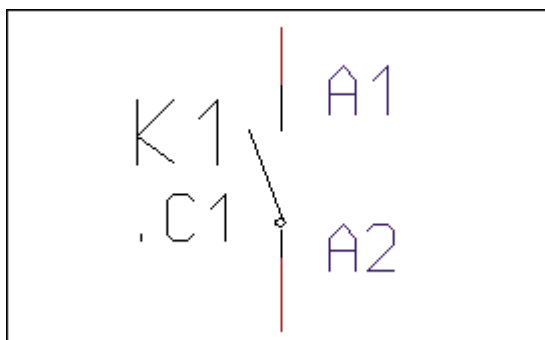
In order to determine the line area, the current page is subdivided into equal sections. The line sections are designated with ascending letters (A - F). The counting direction is from top to bottom.
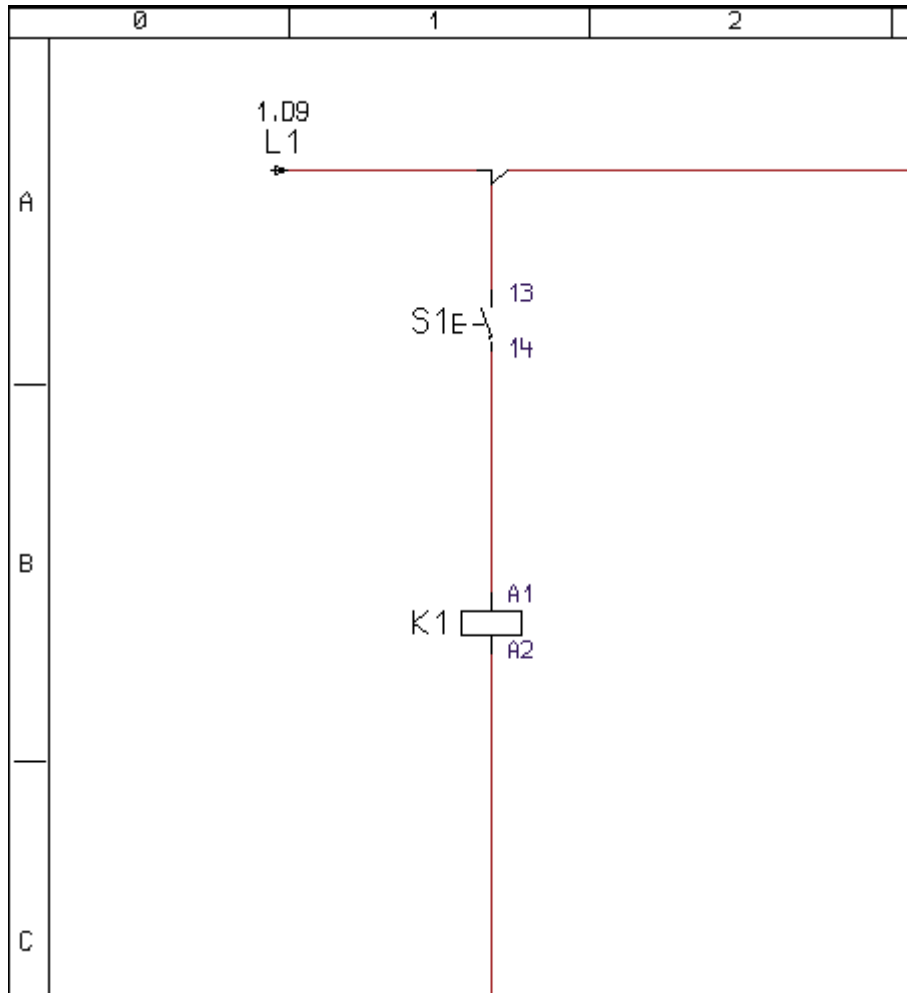
A specific plot frame is required to subdivide the schematic page into line areas. For this reason, the EPLAN installation includes the plot frame `ESSG004D.SKG` for the cross-reference display according to ICE 1082-1.

EPLAN 5 Basic Training

The line division is carried out in the project-specific graphics parameters by making an entry in the Line size input field. For detailed information on this parameter, refer to the section "The Parameters on the Graphics 1 Tab" (User Guide II).

Notes:
The path and line specifications contained in the cross-reference do not have an additional separator.
The specified cross-reference format applies to the schematic as well as to all cross-reference specifications in databases or on generated pages.
There isn't an offset for the desired subdivision. The subdivision starts at the upper zero point and goes to the lower end of the sheet. The beginning of the sheet (upper zero point) is the top edge of the plot frame. The bottom edge of the plot frame represents the lower end of the sheet.

Contactor Selection

Selecting Contactors Off-line

For off-line selection start the procedure after the schematic has been completed. This lets you select all contactors in the entire project in one pass.

Generate > Contactor selection > Contactor > Select

Select the **Contactor > Select** menu items. EPLAN reads the project schematic pages, starts a selection run (based on the parameter settings) for each contactor it finds, and presents the part numbers and type designations of the appropriate contactors or the appropriate combinations of auxiliary blocks in the Contactor selection window.

☞ It may occasionally occur that EPLAN cannot find a suitable contactor, mainly if only a project-specific or general contactor list is used. EPLAN then asks if the search should be continued, suggests all the suitable contactors from the contactor specification file and changes the selection run to the specified selection file for the next contactor.
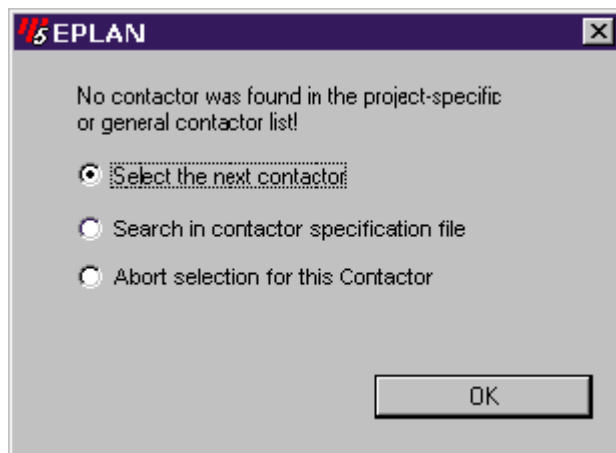
The DD dropdown list box in the upper area of the Contactor selection dialog box lists all the DDs contained in the schematic. The number of the currently highlighted DD is displayed in the Contactor text box; under From: you will find the number of DDs; this however only applies to the off-line selection.

EPLAN 5 Basic Training

The three list boxes below show the technical data for the contactor currently set under this DD.

The window on the left lists the available **contacts** with the coils indicated before the other elements of the contactor. The number of every element within the contactor is displayed (Available) as well as the number of the already used and the unused elements contained in the schematic.

ePLAN® 5

EPLAN 5 Basic Training

The windows in the middle and on the right show the **contact image** of the contactor, the window in the middle for the elements of the contactor, and the window on the right for the elements of an auxiliary block possibly assigned to this contactor.

The upper two window entries are reserved for the **coils** and the other entries for the remaining elements of the contactor or auxiliary block. In addition to its **designation**, the two **connection designations** are displayed for each element.

An element is shown against a gray background in the contact image if it is already used in the schematic. In addition, a red check mark is displayed on the left-hand side of the element if the connection designation in the schematic is identical to the connection designation in the contactor.

Furthermore the coil voltage and the contact rating are displayed for the selected contactor. Depending on the parts program specified (EPLAN parts management or other ERP) these data are taken either from the master parts file of from a transfer file (extension .TDT). This requires that you have entered the contactor in the parts file under a part number that is identical to the part number you entered in the contactor specification file.

You will find a list box below in which you can **select the contactors**. For each contactor (and possibly the associated auxiliary block) the part number and the type designation is displayed. For the part number a variant may be defined (column "Var.") Note that EPLAN also displays combinations of a **single** contactor with various auxiliary blocks.

☞ Please note that you only transfer the contactor for the **current** device by clicking on the **[OK]** or **[Apply]** button. Therefore it is not sufficient to select several contactors and confirm the transfer of all contactors by clicking on **[OK]**.

At the bottom of the dialog you will find a group box. Here, you can **adapt** the **settings** for the contactor selection selected in the parameter management for this selection process. Only the selection file to be used cannot be changed here.

☞ Please note that the two options Minimize unused contacts and Number of unused contacts permitted have no effect in this case. In the case of power contacts, all contacts are offered for selection.
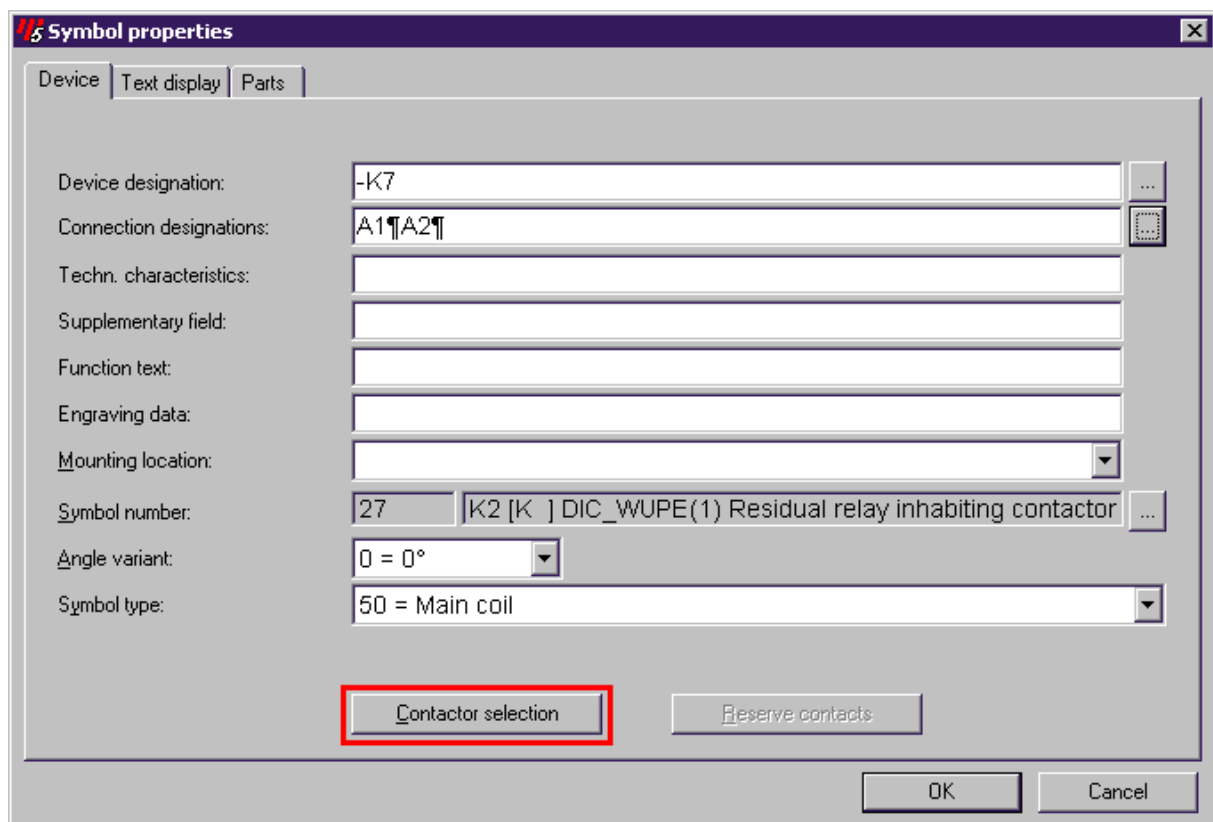
ePLAN® 5

When you click on the **[Display]** button the settings will become effective. Afterwards the list box will be filled with the contactors valid under the selected parameter setting. A warning message may also appear informing that no contactor could be found for these settings.
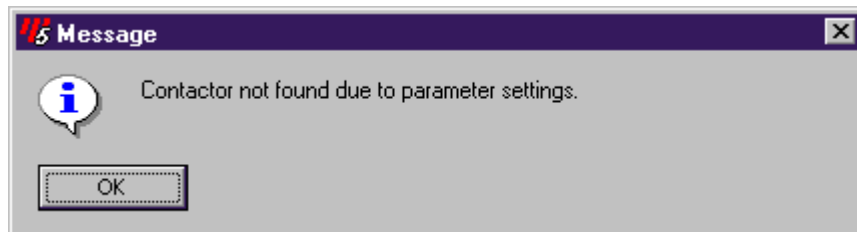
In this case you have to use less "restrictive" parameter settings.

Online Contactor Selection

EPLAN also allows you to select contactors online. That is, while you are creating a schematic. When specifying the properties of the **main coil** select the **[Contactor selection]** button in the Symbol properties window.

EPLAN 5 Basic Training

EPLAN then initiates the selection run for the current contactor and loads the part number, the connection designations, as well as the technical characteristics. You can also use the online process to correct any errors found in individual contactors during the off-line selection.

Using Existing Contacts

Once you have selected a contactor (**[Contactor selection]** button in the Symbol properties dialog box for the main coil) you can have the unused contacts displayed by clicking on the **[Reserve contacts]** button (in the Symbol properties for the other contactor elements dialog box) and select them with the cursor after having highlighted a contact.

Contactor data are **automatically updated online**. This means that not only will the part number and type designation be written back to the schematic, but all connection designations as well. This applies to all pages on which contacts of the contactor being currently edited are located.

Correcting the Contactor Selection

Generate > Contactor selection > Contactor > Correct

You can change individual contactor selections at the end of a selection run without having to repeat the selection process for the whole project. First activate the **Correct** menu item. A list containing the already selected contactors will be displayed.

In addition to the DD, the part number as well as the type designation are displayed in the list for each contactor.

In this list you highlight those contactors that are to be corrected; it is possible to highlight several contactors at once.

After having clicked on the **[OK]** button, change over to the familiar Contactor selection dialog in which you can repeat the contactor selection.

☞ Note that direct assignment of the part number without a check is only possible in the graphic when you input the symbol!

EPLAN 5 Basic Training

Setting Parameters for the Contactor Data Output

 Utilities > Parameters > Project > Contactor/cross-reference/wiring

On the Contactor 2 tab you can specify the parameters that influence the output of the part numbers and of the type designations of the contactors in the schematic. You can access the parameters either from the contactor selection module or via the parameter-management function.

EPLAN 5 Basic Training

These parameters include:

Output identifier:
By entering "1" or "2", you can have the part numbers and the type designations output above or below the contact image in the schematic. If "0" is specified, no number will be output.

Output auxiliary block:
If you activate the check box, the auxiliary block will be output in addition to the part numbers and type designations. In order for this to occur, you must have specified either a "1" or "2" in the Part no. output identifier field.

Font size:
Use the dropdown list box to specify the font size in which the part numbers and type designations of the contactors will be displayed in the schematic.

Alignment:
Here again you use a dropdown list box to specify whether the information will be output left-justified, centered, or right-justified (relative to the path on which the contactor coil is located).

Documenting the Contactor Selection

The Print option offers the following functions for printing the contactor data:

Print contactor specification file
Print selected contactors
Print contactors not selected.

The procedure is the same for all lists. The lists can be output to the printer, the screen, or a file. Refer to the section "Specifying the Output Format for Printing" in the User Guide II.

EPLAN 5 Basic Training

Defining Contactor Lists

Contactor lists are used to limit the contactor selection to a part of the contactor specification file in order to, for example, meet specific customer requirements concerning the types of contactors you can use or to speed up the selection. EPLAN offers two types of lists that you can define and activate after you have selected the **Contactor lists** function from the **Contactor** pulldown menu.

- Project-specific contactor list
- General contactor list.
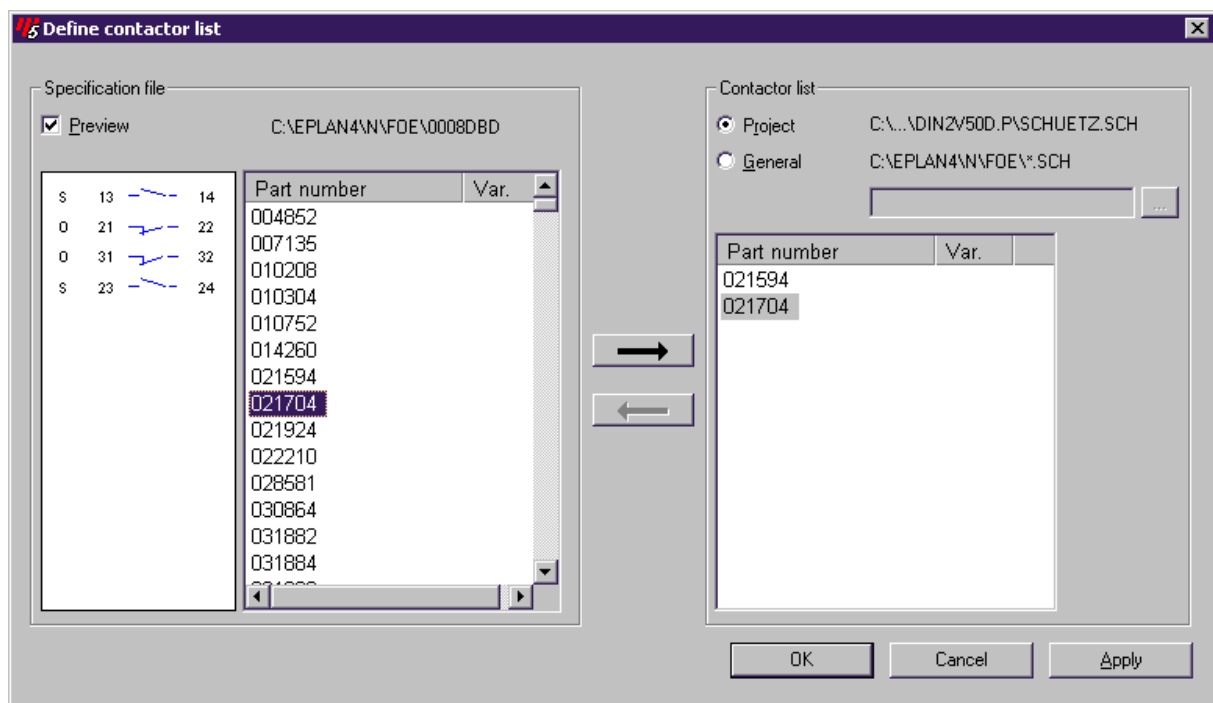
The file name of the project-specific contactor list is a fixed one. You can specify the name of a general contactor list by either entering it in the text field under General or by using the **[...]** button. Enter a new file name here in order to create a **new** general contactor list.

EPLAN 5 Basic Training

You define a contactor list by highlighting data of the contactor specification file in the left-hand area of the dialog box and then copying them to the area on the right-hand side with a click on the [→] button. You can also delete elements from a contactor list by highlighting them in list box in the right-hand half of the dialog box and then clicking the [←] button.

By activating the Preview check box you can have the contact image for the currently highlighted contactor displayed.

By clicking on **[Apply]** the selected contactors are saved in the respective contactor list. In contrast to the **[OK]** button the window will not be closed so that you can select and copy additional contactors or undo the selection.

Defining Contactors

This section describes the structure of the contactor specification file, how you can add and modify contactor specifications, and how to import/export the contactor specification data.

Entries in the contactor specification file are arranged by part number. These part numbers should conform to the part numbers of the corresponding contactors in the master parts file or in the TDT file (depending on the parts management used) so that EPLAN can load the technical data for the contactor from this file.

EPLAN 5 Basic Training

Editing the Contactor Specification File

Select the **Edit** option from the **Contactor specification** pulldown menu. EPLAN opens a dialog box in which you can enter the following information:

EPLAN 5 Basic Training

Part number:
The part number is the key field for the contactor specification file, that is EPLAN uses the part number as the search criterion for contactor specifications. Therefore, each part number must be unique and must be the same as the part number for the corresponding contactor in the master parts file. (Among other things, the master parts file manages information such as coil voltage and contact rating. In other words, technical data for the contactor.) If the two numbers do not correspond or if no number has been included in the master parts data, the bill of materials cannot be generated correctly!

Type designation:
Like the part numbers, the type designation must be unique.

Contactor type / Group:
The identifier in this field specifies the type of contactor in question and the various possible contactor/auxiliary block combinations.
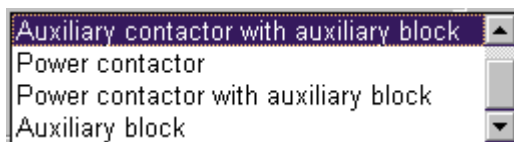
Any three alphanumerical characters can be entered for the group. This way, you divide the contactors into groups so that you can define possible combinations of contactors and auxiliary blocks.

💡 The following contactors were entered in a contactor specification file:

Part No.	Contactor type	Group	Explanation
AAA	Aux. contactor (aux. block)	X	Auxiliary contactor onto which an auxiliary block can be added. If the number of contacts is insufficient, this contactor is displayed together with the auxiliary block DDD for selection (grouping X).
BBB	Aux. contactor (aux. block)	Y	As for AAA, but can be combined with the auxiliary blocks EEE and FFF.
CCC	Auxiliary contactor		This contactor cannot be combined. Grouping is therefore not possible.
DDD	Auxiliary block	X	
EEE	Auxiliary block	Y	
FFF	Auxiliary block	Y	

Part type:

Use this field to specify the part type under which the contactor or relay will be managed in the master parts data. You can define the part as either an individual part ("I") or as an assembly ("A").

Variation:

This field is used to define multifunction relays. Since these relays are all listed under the **same** part number but can have a variety of contact configurations, this field is used to differentiate between the individual variants so that EPLAN can also include multifunction relays in the contactor selection process. EPLAN can then also use multi-function relays during contactor selection.

Coil 1 / Coil 2:

Use these fields to define the coils for the contactor or relay, as well as the corresponding upper and lower connection designations. By using the dropdown list box you can select the symbol names of the coils from the symbol file specified in the parameters under Symbol file.

KM2	Multifunctional relay
KMU	Multifunctional relay
KR	**Relay coil of a slow-releasing relay 07-15-07**
KR2	Relay coil electric drive with dropout delay 07-
KRM	Relay coil of a remanent relay 07-15-19
KS	Latching relay
KT	Counter

☞ Note that during contactor selection EPLAN compares **symbol numbers** of the coils. In other words, if it encounters the coil symbol "KT" in the schematic, EPLAN will also search the contactor specification file for a coil designated as "KT". If you have not defined a coil with this designation in the contactor specification file, EPLAN cannot find a matching contactor. You should therefore take care not to make any incorrect entries when making a manual entry.

Contacts 1 to 12:

Use these fields to define the contact configuration of the contactor or relay as well as the contact's connection designations.

Note that during contactor selection EPLAN compares contacts by the **symbol number** and **symbol type**. For example, for the contact symbol "SL" (normally open power contact) with a symbol type of "0" (normally open power contact) in the schematic, EPLAN will look for a contactor with an "SL" contact with symbol type "0" in the contactor specification file.

If you have not specified a contactor with this contact in the specification file, EPLAN cannot find a matching contactor. Note that while the symbol "S" (normally open) and symbol type "0" also represent a normally open power contact, this designation is not the same as "SL" and "0". Therefore ensure that you only enter valid symbol names when entering them manually, and that you adapt the symbol type of the contact.

EPLAN 5 Basic Training

Symbol type:
In the **Symbol type** column you can have the symbol types of the entered coils and contacts displayed and check them.

☞ If a specific contact is to remain free assign the **first** entry (blank entry) from the dropdown list box to it.

The individual **command buttons** of the dialog have the following effect:
[
[New]:
This function lets you re-create contactor specifications. The data of the contactor last selected are retained as default values and can be revised.

[Change]:
Use this button to accept changes.

[Blank entry]:
Use this button to empty all fields in the dialog box. The previously displayed field of the contactor specification file remains unchanged. After having filled the fields you can transfer the entries by clicking on the **[New]** button.

[Delete]:
This function allows you to delete the record displayed in the Contactor specification window after a prompt for confirmation.

[Close]:
Use this function to close the dialog box. A prompt for confirmation is displayed if you have not yet saved the changes you made in the current record.

☞ If you have specified new contactors or renamed existing contactors in the contactor specification file, you should also enter the corresponding records for the new contactors in the parts master data.
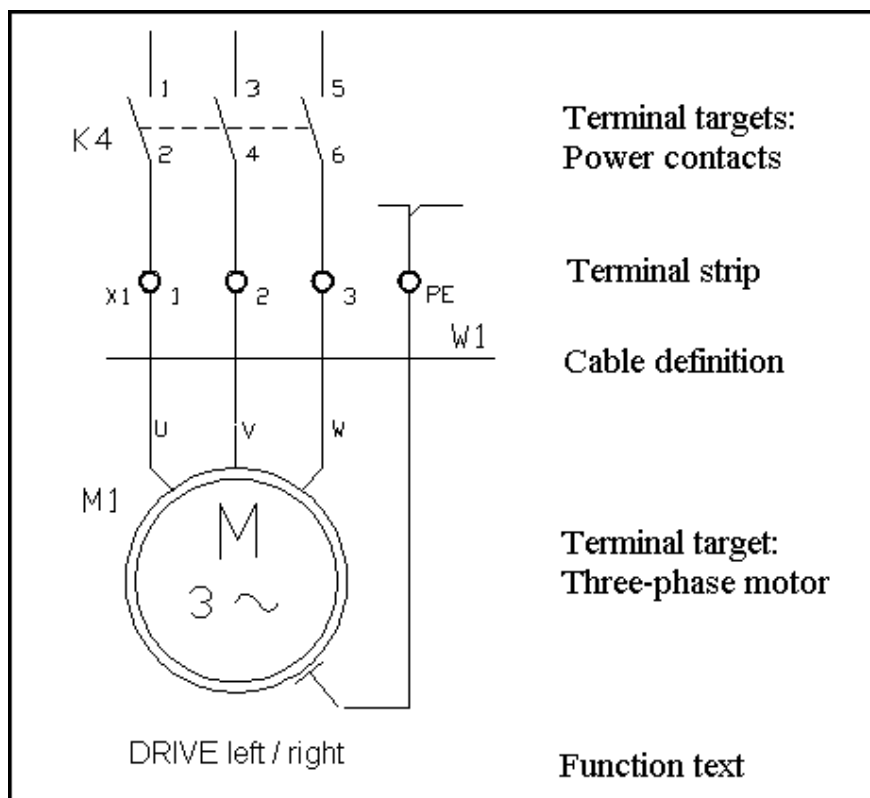
EPLAN 5 Basic Training

Terminals

The next sections will provide you with information on the following topics:

- Inserting Terminals
- Multi-level Terminals in EPLAN
- Defining Terminal Properties
- Inserting PLC End Terminals.

Inserting Terminals

Terminals are considered to be devices and are inserted from the symbol file in the same way as general symbols. The symbol name always begins with an x. Note that EPLAN only evaluates these terminals for the terminal diagram. (However, the evaluation programs recognize the terminal properties from the information on the symbol type, not from the symbol name.)
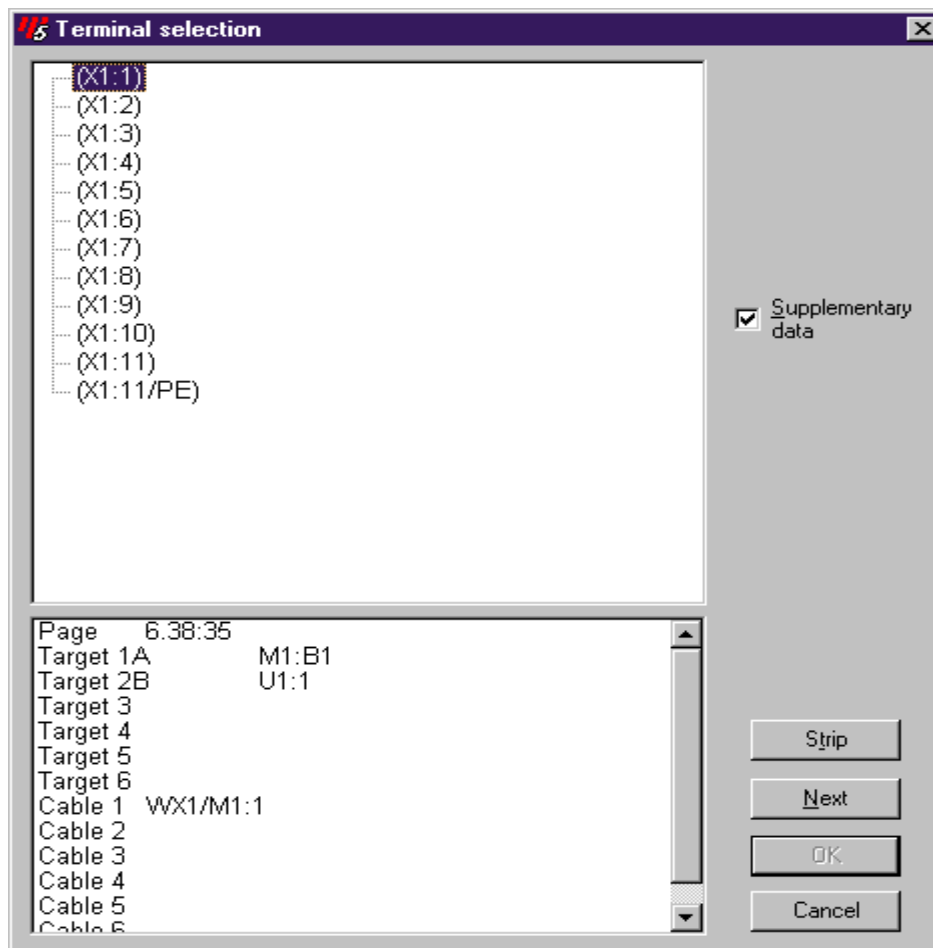
EPLAN 5 Basic Training

Terminals are added in the same way as "normal" devices. Here too, a list window is displayed from which you select the desired terminal, whereby the terminal's graphic appearance is shown in a preview window on the screen.

When you have inserted a terminal at the desired position by single-clicking it, a dialog box opens in which you have to enter the terminal's properties. You can subsequently change these properties by positioning the cursor on the terminal's insertion point and pressing ⏎ or the left-hand mouse button.

Since terminals and terminal strips are managed by EPLAN in a project-specific database, you can use the system-supported terminal assignment function (**[...]** in the Designation input field) to have the existing data displayed and to assign new terminal numbers.

EPLAN 5 Basic Training

Note that an up-to-date terminal and interconnect diagram must exist!
In contrast to "X" terminals, device-end terminals that you add by either pressing
⬚ ⬚ or activating the appropriate icon in the Insert toolbar are not treated as devices. Instead, these terminals merely represent connections for devices and are therefore not shown in the terminal diagram (cf. the section entitled "Inserting Device-end Terminals").

Multi-level Terminals in EPLAN

Multi-level terminals are used when specific control panel or mounting panel measurements must not be exceeded and thus a space-saving wiring is required.

A large number of terminal types is offered by the manufacturers which differ in the field of application and in the designation. A rough difference is made between multi-level terminals with independent levels and multi-level terminals with connected levels.

Multi-level Terminals with Independent Levels

This terminal type includes several terminals that lie one upon the other but have no conducting connection.

The names usual in trade are:

- Double-level terminals
- Installation terminals
- Initiator terminals
- Actuator terminals
- Shield terminals
- Three-wire terminals etc.

A special form of these terminals integrates all connections of an initiator (e.g. proximity switches) or actuator (e.g. valve coil). This results in an extremely high wiring density. The highest level is used as passage for the signal line, whereas, for example, on the middle level the minus potential and on the lowest level the plus potential of the initiator power supply is distributed over snappable insertion bridges.

ePLAN®**5**

EPLAN 5 Basic Training

Multi-level Terminals with Connected Levels

A multi-level terminal with connected levels is a terminal that disposes of more than two physical screwing possibilities. Since, according to DIN, two conductors can be clamped under each screw, up to 12 conductors can be wired to a three-level terminal (3 levels x 2 screws/level x 2 conductors).

The names usual in trade are also:

- Distributor terminal
- Shunting terminal
- Three-wire terminal
- Potential connector etc.

In EPLAN, multi-level terminals are defined by using the terminal properties (see next chapter).

The special text 450 ("Levels") is available for the graphics output of multi-level terminals into the terminal diagram. This allows you to define the level as well as the jumper.

The following table provides an overview:

Definition in form	Output		Remark
Special text 450	Special text 400	Special text 450	
Not available	As before		No change to existing outputs
Value = 0	As before	Output of level as numeric value	"0" corresponds to the "standard terminal"
Value <> 0	All wire jumpers (same position as before)	All jumper bars (open circle, if no jumper)	To graduate the circle diameter more finely, the double diameter for the font in relation to the wire jumpers must be selected!

ePLAN® 5

Defining Terminal Properties

The properties of the terminal are defined in the Symbol properties dialog box. That is, you determine how EPLAN will evaluate this terminal for display in the terminal diagram.

☞ Please note that specific fields in the Symbol properties dialog box cannot be displayed or edited when you insert terminals / connectors from a "single-pole" symbol file into a schematic page of the "X = Single-pole display" type.

The following input possibilities are available on the Terminal / connector pin tab:

Designation:
Enter the terminal's device designation here. The designation can consist of the higher-level assignment, location designation, terminal number, and terminal subnumber. You can use the **[...]** button located next to the Designation field to assign terminal designations system-supported.

Connection of the external output position in form:
In this entry you define the direction that is considered to be "external" when viewed from the terminal in the schematic. The term "external" is used to specify a specific location on the terminal diagram form. The terminal targets found by the EPLAN search in the specified direction will be entered on the external side (position "A" and "C") on the terminal diagram.

Connection for cable generation:
In this entry you specify in which direction from the terminal in the schematic cables will be generated automatically.

☞ The automatic cable generation is not possible on single-pole pages!

Utilities > Parameters > Project > Terminals/cables

The cable that will be used is defined by your entries in the Standard cable type, Standard number of conductors, and Standard conductor cross-section fields of the project-specific terminal-diagram parameters.

EPLAN 5 Basic Training

PLC designation:
If you use a terminal-diagram form which provides for the PLC designations to be output, the text entered here will be output.

The text in this field can be translated online. The online translation function (see the chapter on "Translating Texts Online" in the User Guide II) is started by pressing the right-hand mouse button to call up the **Foreign languages** pop-up menu.

EPLAN 5 Basic Training

Function text:
Enter a function text of up to 255 characters for this terminal into this field. This text is output in the "Function text" column of the standard terminal diagram. The function text entered here can also be translated online.

Symbol number:
This field indicates the internal position of the symbol in the symbol file. Click on **[...]** to change the current selection.

Angle variant:
This dropdown list box shows the existing angle variants for the current terminal.

Note that a change of this setting has an influence on autoconnecting!

Symbol type:
Enter the symbol type for the terminal here. The symbol type is evaluated during the generation of the terminal line-up diagram. The type is used to differentiate between the individual terminal types.

You can also launch the Selection of the terminal symbol type dialog box via the **[...]** button and select the symbol type of the terminal there. The section "Symbol Type: Terminal and Accessories" of the System Manual provides an overview of the terminal symbol types.

Multi-level terminal on level:
If you activate this check box after having entered a "normal" terminal, this terminal is defined as multi-level terminal. When specifying the level you determine the clamping point in the terminal housing, thus the story or level.

☞ If you select a level > 1 the Part tab disappears from the terminal properties dialog box.

ePLAN® 5

Terminal strip search direction:
Use the appropriate identifier to enter the direction in which EPLAN should search to find the strip designation for this terminal.

Sorting identifier:
Use this identifier to specify how the terminal is to be sorted into the terminal diagram. You can open the Terminal positioning dialog box via the **[...]** button, where the position of the terminal on the associated strip can be changed.

Check identifier:
Use this identifier to specify whether the terminal may occur only once or several times in the terminal diagram.

Jumper bars:
By entering the appropriate identifier in this field, you can specify whether you want to define jumper bars for this terminal, regardless of the automatic evaluation. The possible entries can be selected from the dropdown list box.

In addition to the terminal properties you can specify the display of the terminal number or of the terminal-strip designation in the properties dialog box. This is done by using the Text display tab, as is the case for "general" symbols from the symbol file (see the section on "Defining the Display of the Symbol Texts" in this chapter).

Assigning Terminal/Connector Designations System-supported

Just like device designations, terminal or connector designations can be assigned system-supported, too.

To do so, highlight the terminal or connector of any strip in the schematic, or insert a new terminal or connector into an existing strip.

EPLAN 5 Basic Training

Afterwards click on the **[...]** button located next to the Designation field in the Symbol properties dialog box for terminals or connectors (Terminal / Connector pin tab).

The Terminal selection dialog box that comes up provides an overview of all terminal designations/connector designations used in the project for the terminal strip/connector strip highlighted in the schematic or to which you have added a terminal/connector in the schematic. Entries in round brackets signify that the corresponding terminal/connector is actually used in the project. Entries without brackets indicate spare terminals or spare connectors.

If you activate the **Supplementary data** check box, the cross-reference data and connection targets for each terminal strip/connector strip that has been assigned are displayed in the lower window.

Click on **[Strip]** to open the Strip selection dialog box which gives an overview of all terminal strips/connector strips used in the project. Here you can select the strip which you wish to assign to the inserted terminal/connector.

If you click the **[Next]** button, EPLAN will automatically suggest the next available terminal number/strip number.

Confirm the suggested designation by clicking **[OK]**; it is then entered into the Designation field of the Symbol properties dialog box. Finally click **[OK]** again, and the new terminal/connector designation is read into the schematic.

☞ You should already have generated a terminal diagram beforehand in order to ensure that the terminal database has been updated and is error-free.

ePLAN® 5

EPLAN 5 Basic Training

Insert PLC End Terminals

PLC end terminals are used to display the circuit of sensor or actuator components at PLC assemblies in the schematic. In EPLAN, PLC end terminals are used on pages of the page type "Q = PLC page" as well as on pages of the page type "T = PLC overview". Insert PLC end terminals on pages of the "Q" type to realize the byte-wise connection assignment of PLC inputs and outputs of an I/O board. This is different for page of the "T" type: you will need them here to create a board-oriented complete overview of the connection assignment of all PLC input or output boards.

👉 In EPLAN COMPACT you can also insert PLC end terminals in project pages of the type "Q = PLC page" that have been created with the full version. However, due to the limited functions compared to the full version, EPLAN COMPACT does not allow to evaluate PLC end terminals during generation runs.

In general, you must insert PLC end terminals into a black box to make PLC assignments visible on T or Q pages. On PLC overview pages this black box is inserted as "PLC box".

👉 Note that after having inserted the black box into a PLC page you must set the symbol number to "-13 = I/O boards" in the Symbol properties dialog box. For detailed information on how you insert black boxes into the schematic in EPLAN, refer to the section "Inserting Black Boxes" in this chapter.

Now insert a PLC end terminal into the black box (or the PLC box) by following the menu path **Insert > PLC end terminal** in the graphics editing module.

When the terminal has been placed, the Symbol properties dialog box is opened. In the PLC end terminal tab specify the properties of the inserted terminal which EPLAN requires for generating PLC cross-references.

ePLAN® 5

EPLAN 5 Basic Training

Entering Symbol Properties for PLC End Terminals

The following entries can be made in the PLC end terminal tab:

Address:
Enter the full address of the PLC end terminal here. The address usually consists of an identifier which marks the connection as input or output, a number for the byte address, a separator, and a number for the bit address, e.g. O8.4 (Output, Byte 8, Bit 4).

☞ By means of the PLC generator you can edit PLC addresses and write them back to the schematic.

As an alternative to manual input you can also click the **[...]** command button to make a system-supported address selection. EPLAN then displays all the PLC addresses still available in the subsequent PLC address selection window. Which entries the selection list contains, depends on the respective insertion situation:

Situation 1
You insert the PLC end terminal into an I/O board (black box with symbol type "-13"), which already contains **a device designation**. In this case, the list in the selection dialog box only shows the free addresses of the I/O board currently edited. Select the desired address and confirm it by clicking **[OK].** The address will then be written to the PLC end terminal.

Situation 2
You insert the PLC end terminal into an I/O board which does **not** contain **a device designation**. The list of the selection dialog box then shows the free addresses of **all I/O boards used in the project**. When you highlight an address in the selection dialog box and confirm with **[OK]**, EPLAN takes over the device designation of the I/O board contained in the address and writes it to the I/O board currently edited. Please note that no DD is taken over at the I/O board if you manually change the address after a system-supported selection has been made.

ePLAN ® **5**

EPLAN 5 Basic Training

Situation 3
You insert the PLC end terminal outside an I/O board. The list of the selection dialog box then also shows the free addresses of **all I/O boards used in the project**. Once you select an address in the selection dialog box and confirm the dialog box with **[OK]**, EPLAN takes over the selected address at the currently inserted PLC end terminal without considering the device designation.

👉 If you want to change the symbol properties of several PLC end terminal at one go by using the block editing function, it is not possible to make a system-supported PLC address selection.

Corresponding address:
Numerous PLC manufacturers allow to feed sensors directly via operating-voltage-carrying terminals which are also contained on the I/O board. You can specify a so-called "Corresponding address" for these supply terminals to assign them to the signal-transferring (addressed) PLC terminal. This field can only be edited if you have selected Power supply for the Type of connection!

Connection description:
This field is used to enter a connection designation for one of the following connection types:

- Not specified
- Digital input
- Digital output
- Analog input
- Analog output
- Board power supply
- Bus

This field cannot be edited if the Power supply type of connection is selected!

EPLAN 5 Basic Training

Type of connection:
You can select among various types of connection for a PLC end terminal. The following options are available:

- Not specified
- Digital input
- Digital output
- Analog input
- Analog output
- Power supply
- Board power supply
- Bus

Connection subtype:
In addition connection subtypes can be selected for the Power supply, Board power supply, and Bus types of connection.

In case of Power supply and Board power supply you can determine whether the connection has a plus potential, minus potential, or PE conductor potential, if it is intended for the connection of a shielding or if it is not specified at all.

In case of Bus you can decide whether the connection is an input or an output or is not specified at all.

Symbolic address: (not available on PLC overview pages)
If necessary, enter the symbolic address of the PLC end terminal here. Symbolic addresses may contain symbolic names. This makes it easier to identify PLC connections, e.g. "FEEMED2" for "Feeder Medium 2".

Supplementary field: (not available on PLC overview pages)
Use this field to enter general information for the PLC end terminal. This information is displayed and plotted if the Show supplementary field parameter is activated under **Utilities > Parameters > Project > Graphics, Graphics 4 tab**.

The text in this field can be translated online. The online translation function (see the chapter on "Translating Texts Online" in the User Guide II) is started by pressing the right-hand mouse button to call up the **Foreign languages** pop-up menu.

ePLAN® 5

EPLAN 5 Basic Training

Function text: (not available on PLC overview pages)
A function description for the PLC end terminal is entered in this field. The text you enter here, which may contain 255 characters, takes precedence over the function text in the same path, which you can enter as a Function text text type. This information is displayed and plotted if the Show function text project parameter is activated (**Utilities > Parameters > Project > Graphics > Graphics 4 tab**).

The function description entered here can also be translated online.

**Terminal display:** (not available on PLC overview pages)
You can select between the following display modes in this dropdown list box:

- **With graphics:** A terminal with a connecting point is drawn. If required, this terminal is also shown in a black box.
- **Without graphics, designation lying outside:** Only a connecting point is specified. The wiring ends at this point. If you place this terminal inside a black box, the connection terminates at the edge of the box. The designation of the device-end terminal appears outside the black box.
- **Without graphics, designation lying inside:** Same as above, but the terminal designation appears inside the black box.

Connection designation: (only available on PLC overview pages)
Using this field you can specify the connection designations or numbers (pin numbers) of the PLC end terminals on the I/O board.

Connection direction:
After the **Insert > PLC end terminal** or **Insert > Device-end terminal** function has been selected, keep the Ctrl key pressed down and move the mouse up, down, right, or left to rotate the connection direction. The original connection direction is displayed in this field and can be modified.

By means of the property dialog box you can not only influence the PLC end terminal properties but also the display of the PLC addresses. This is done by using the Text display tab, as is the case for "general" symbols from the symbol file (see the section on "Defining the Display of the Symbol Texts" in this chapter).

☞ Notes:

To generate a cross-reference between schematic page and PLC overview page, the address and type of connection/connection subtype as well as the device type must be identical.

A function text (Function text text type) can be specified at a PLC end terminal when a PLC end terminal is inserted as well as when a text is inserted in graphics editing. The function text for the PLC end terminal takes precedence during the evaluation.

The connection designation entered on the PLC overview page at the PLC end terminal can be written back to the PLC page by selecting the menu path (from the main menu) **Generate > PLC generator > Generate > Write connection numbers back to schematic**.

The function text as well as the symbolic address can be displayed on PLC pages by opening the PLC generator module and then selecting **Parameters > PLC data**.

EPLAN allows you to generate symbolic addresses by means of the PLC generator. You can decide whether a DD target tracking is to be carried out.
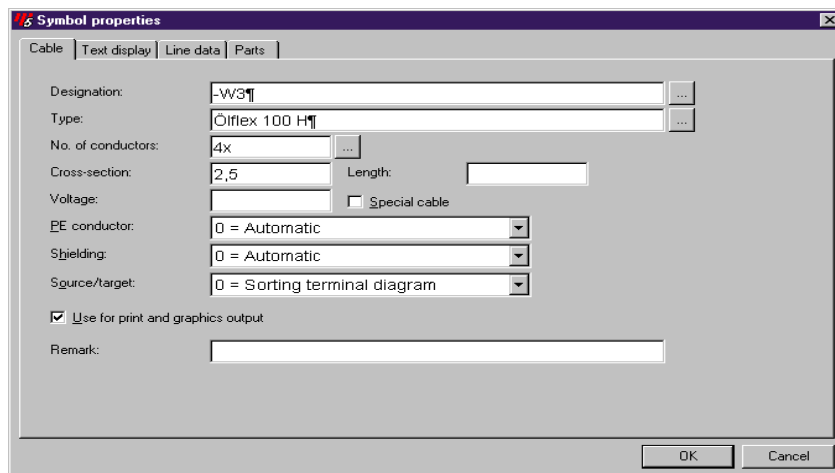
EPLAN 5 Basic Training

Cables

In EPLAN you can define the characteristics of cable definitions and of the wiring, which will be used during the evaluation runs.

Use the Cable definition function to add specific cable definitions to the schematic by hand. These definitions deviate from the general entries you made in the parameters program. Definitions you add in this way will then be evaluated in both the terminal and interconnect diagrams. In addition, you can also define a cable between two symbols that are no terminals. Proceed as follows to define a cable:

- Press ⬚ F5 , or click the ⫿ icon in the Insert toolbar, or select the **Cable definition** option in the **Insert** pulldown menu.
- Use the mouse to specify the desired line.
- Enter the corresponding data in the Symbol properties dialog box.
- Finally, use the mouse to dynamically specify the text position.

Input Fields for the Cable Definition

Enter the cable-specific data, for example the cable designation, cable type, etc., in the Cable tab. The following fields are available:

EPLAN 5 Basic Training

Designation:
Enter the cable designation into this field. When editing in accordance with DIN, you can assign higher-level assignments and/or location designations by using "=" and "+" as prefixes. The cable designation may consist of both numbers and letters. If the cable designation does not start with a letter, the identifying cable letter defined in the terminal-/cable-specific Cable identifier parameter is used. If you do not enter anything in this field, the cable designation will be generated automatically for the terminal or interconnect diagram. In this case the cable designation will be based on the cable designation defined in the terminal/cable parameters.

The cable designation can also be assigned with the support of the system. For detailed information refer to the following section "System-supported Assignment of Cable Designations".

Type:
The cable type defines what kind of cable is being used. This is important when assigning color codes and when the maximum number of conductors is considered. If color codes are to be assigned or if there is a maximum conductor number, the cable file must have an entry containing the name specified here and the corresponding data. Use **[...]** to access the data already created.

No. of conductors:
If the form used contains a corresponding entry, the number of conductors entered here will be imported into the terminal and interconnect diagrams. Enter the number first. You can then enter an "x" as a separator between the number of conductors and the cross-section. You must enter this "x" if you are using the standard forms supplied by EPLAN Software & Service. In order to specify conductor pairs, you can enter a second "x". If you do not want to use the unit of measure for the cross-section that is specified in the parameters, enter the "x" separator as a capital letter.

The number of conductors specified for the respective cable type can be viewed, and applied, by clicking on **[...]**. The "x" separator is then added automatically by EPLAN.

EPLAN 5 Basic Training
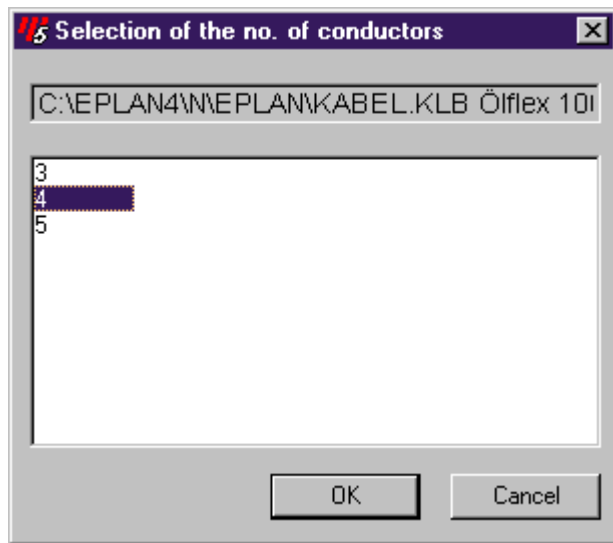
Cross-section:

If the form used contains a corresponding field, the cross-section entered here will be imported into the terminal and interconnect diagrams. The maximum cross-section is 655. If the cross-section of a PE conductor is different to those of the other conductors, you can use a "+" as a separator and thus enter two different cross-section values (e.g., 150+80mm^2).

☞ Keep the no. of conductors in mind for this specification. If, for example, you entered a "4" into the No. of conductors field, and the entire entry is therefore 4 x 1,5+2mm², the cable does not dispose of 4 but of 5 conductors!

Length:

The value you specify in this field will be used during cable definition in the terminal or interconnect diagram if a corresponding field exists in the diagram form. The length can be entered with decimal values, or you can enter a unit in this field. An entry depends on the type of form you are using. If you are using the standard forms supplied by EPLAN Software & Service, you must define the unit of measurement. If you do not specify a particular unit of measure, the value specified in the corresponding parameter will be used.

EPLAN 5 Basic Training

Voltage:
The value for the voltage is entered in this field. It is not evaluated at present.

Special cable:
Use this check box to specify whether the cable is to be evaluated as a special cable. This means that if the data of this cable differ from those of a cable of the same type and this check box is activated, the "Differing cable data" message is suppressed.

PE conductor / Shielding:
Use the dropdown list box of these fields to specify whether the respective value in the cable type file is to be used or whether you want to specify a different value. There are the following selection possibilities:

- **Automatic:** The definition of the PE conductor / shielding is imported from the cable type defined in the Cable type input field.
- **Without:** Even if the cable defined under Type provides for a shielding or PE conductor, it is not created in the interconnect diagram.
- **1 PE / One-end:** If this entry is selected, a shielding / PE can be generated.
- **2 PE / Both ends:** Select the 2 PE entry if two PE conductors are to be allowed for the cable. If the shielding is connected to both ends, this entry suppresses the message "Two/several shielding conductors intersected".

Source/target:
Use this entry to specify whether the target designations in the interconnect diagram should correspond to those of the terminal diagram or whether the target designations are to be exchanged.

Use for print and graphics output:
If you activate this check box EPLAN will consider the cable specified here and includes it in the printout as well as in the graphics output of the interconnect diagram. This is the default setting. If the check box is deactivated the cable is ignored during output.

Remark:
You can use this field to enter additional information about the specified cable definition. The maximum length is 80. You can use any character except control codes.

EPLAN 5 Basic Training

The entries made in the Designation, Type, Length, No. of conductors, Cross-section, Voltage, and Remark fields are displayed as cable definition texts on the schematic page once the dialog box has been closed. If you want to display the cable definition texts in several lines you must enter a line feed behind the respective field entry. To do this, press the key combination [Ctrl] [↵] or select the Line feed option from the pop-up menu after you have entered the text. In the text field, a line break is represented by a "¶" symbol.
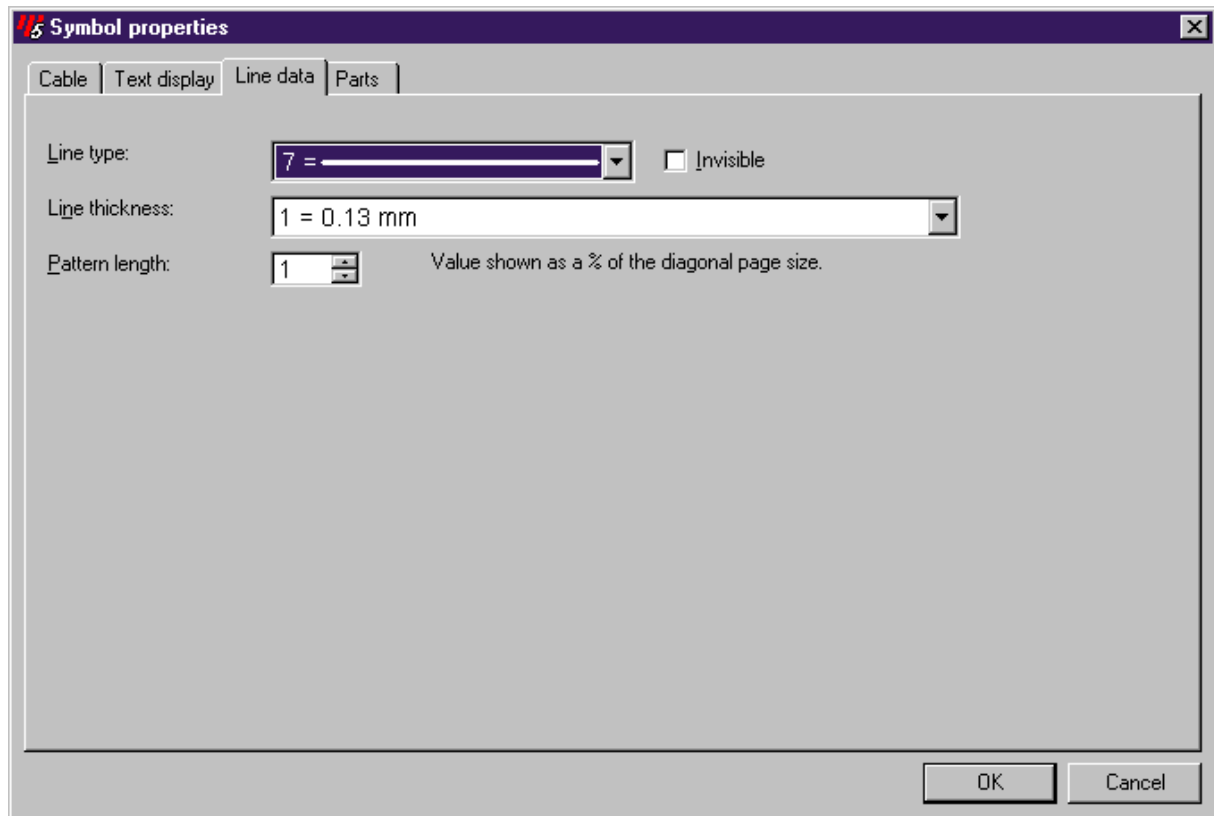
Text Display of the Cable Definition

In addition to the cable properties, you can also influence the display of the cable definition texts in the properties dialog box. This is done by using the Text display tab, as is the case for "general" symbols from the symbol file (see the section on "Defining the Display of the Symbol Texts" in this chapter).
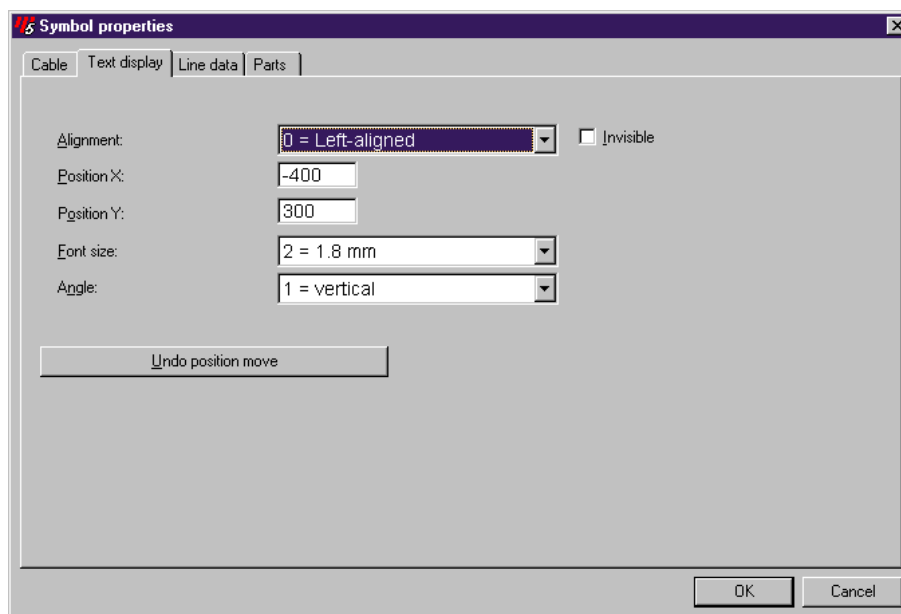
Line Data of the Cable Definition

You can determine the line type and thickness as well as the pattern length for the actual line in the Line data tab (see the section on "Defining Line Data of Black Boxes and Wirings" in the User Guide I).

👉 The extension of an already defined cable line can also be subsequently edited by calling up the pop-up menu on the insertion point of the line and activating the **Change size** option.

EPLAN 5 Basic Training

Parts Data of the Cable Definition

Finally enter the part-specific information concerning the part number, number of units, etc. in the Part tab. Use the **[Select]** button here to branch to the parts management function and access the data saved there.

	Part type	Part number	Contactor	Qty.	Part cla	Type of	Subgroup
1	5 = Cable	LAPP.0014 104 (4G2		1			
2							
3							
4							
5							
6							
7							
8							
9							
10							

[Delete] [Select]

[OK] [Cancel]

Displaying Cable Properties

The most important cable properties are displayed if you place the cursor on the insertion point of the cable line, right-click to call up the pop-up menu, and select the **Display properties** menu item.

Note that the Cable information window is only a display window, i.e. the cable-specific data cannot be edited here!

System-supported Assignment of Cable Designations

Apart from the manual assignment of cable designations, EPLAN also offers a system-supported solution to designate defined cables.

If you want to use it click on the selection button (**[...]**) next to the Designation field in the symbol properties dialog box for the cable. EPLAN then opens the Cable selection dialog box.
This dialog box provides an overview of all cable designations assigned in the schematic that have been defined by means of a cable definition line in the graphics.

EPLAN displays the cable properties of the currently highlighted cable in a separate field below.

☞ Wires that do not contain a cable definition line but that have been generated by the automatic assignment of the cable designation are not displayed in this dialog box.

There are two alternatives for the system-supported cable designation:

1. Highlight a designation listed in the selection dialog box, and confirm by clicking **[OK]**.
2. You want to use a designation that is not contained in the overview. To do so, click the **[Next]** button. EPLAN then automatically suggests the next designation that has not been assigned yet.

After you have confirmed with **[OK]** the selection dialog box is closed and the selected or suggested designation is entered into the Designation field of the Symbol properties dialog box. Finally click **[OK]** again, and the new designation is read into the schematic.

EPLAN 5 Basic Training

Defining Shieldings

The Shieldings function lets you add the graphics symbol for the shielding of wirings. You can use the mouse to define the size of the shielding. You must activate this function next to the wires for which you want to add a shielding. You can use the mouse to expand and rotate the shielding as required.

Shieldings can also be created by clicking the ⬚ button of the Insert toolbar.

The designation for shieldings entered in the terminal and cable parameters (menu path: **Utilities > Parameters > Project > Terminals/cables**) is evaluated in the terminal or interconnect diagram as the target designation.
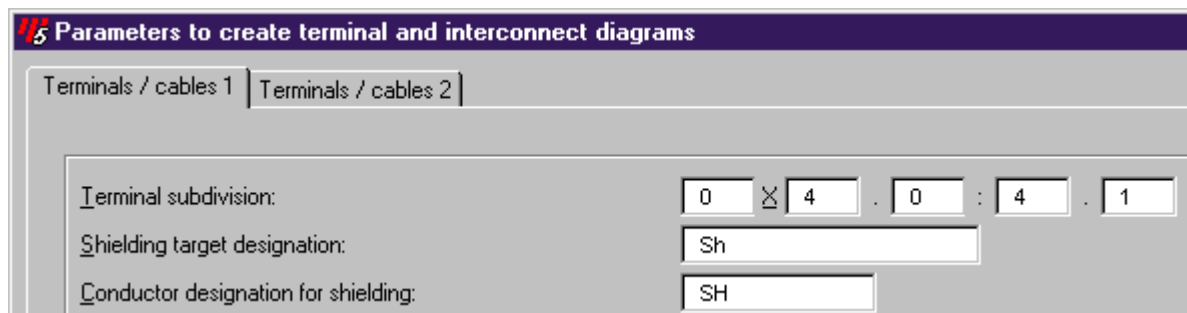
In order to subsequently change the width or height of the shielding, position the cursor on the shielding's insertion point and press ⬏ or the left mouse button. You can then use the mouse to dynamically change the size.

To modify the graphics data of a shielding right-click on the insertion point of a shielding to call up the pop-up menu and select the **Attachment line type** option. Afterwards left-click on the insertion point of the shielding and modify the current data for line type, line thickness, and pattern length.

Creating a Table of Contents

EPLAN allows you to enter page-specific information, for example for the page designation or the supplementary page fields, on every page of your project. The function for the table of contents creation is used to read out these data and present them in the form of separate overview pages.

💡 Extract from a table of contents

Table of contents

Page	Page designation
=DOKU/1	title sheet
=DOKU/2	Table of contents
=DOKU/2.1	Table of contents
=DOKU/2.2	Table of contents
=DOKU/3	EPLAN News 5.50
=AP+0/1	plant overview
=AP+0/2	plant overview
=AP+ST1/1	supply
=AP+ST1/2	drive
=AP+ST1/3	control voltage
=AP+ST1/4	control voltage

The next sections will provide information on the following topics:

- Setting Graphics Output Parameters for the Table of Contents
- Creating a Table of Contents
- Updating a Table of Contents
- Editing Special Text Coordinates in the Table of Contents Form

EPLAN 5 Basic Training

Setting Graphics Output Parameters for the Table of Contents

Various graphics output parameters allow you to adapt the format and the type of output of the table of contents to suit your requirements. The following options are available:

Using a Plot Form

To output the table of contents in project pages you must specify a corresponding plot form using which you can influence the format of the table of contents.
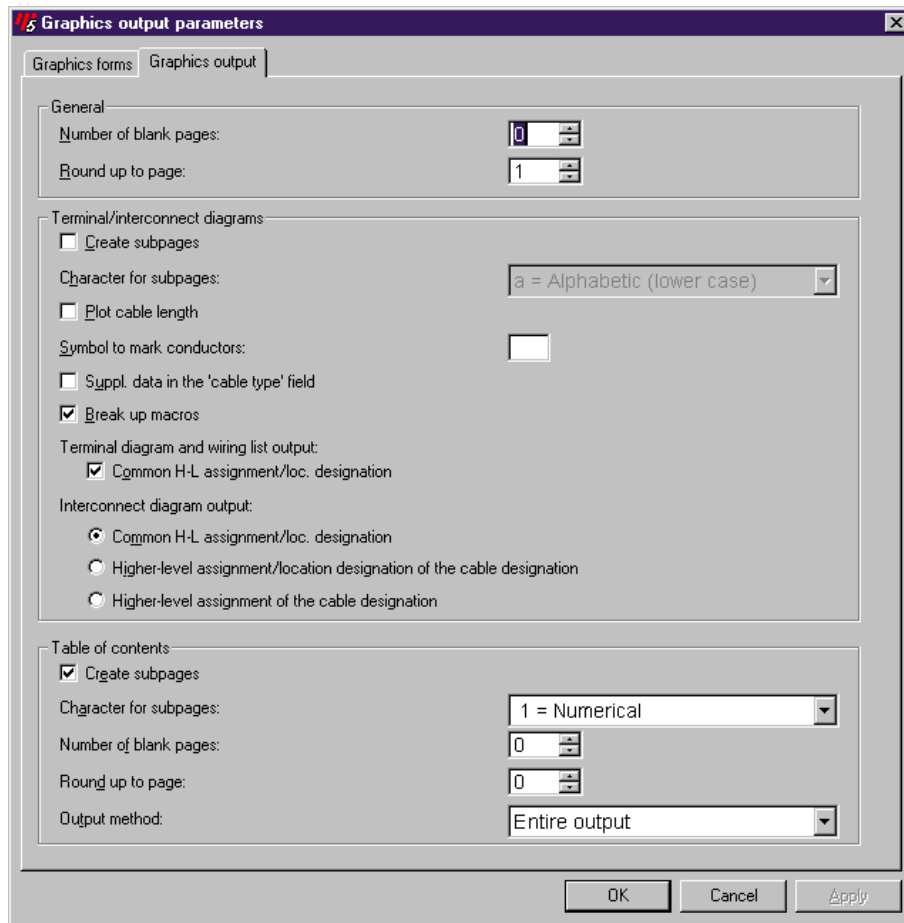
Starting from the EPLAN main menu, follow the menu path **Utilities > Parameters > Project > Graphics output > Graphics form tab**.

Enter the name of the desired plot form into the **Table of contents** field, or select the plot form via the **[...]** button. The forms have the extension `*.SKJ` and are saved in the directory `EPLAN4\N`. The check box in front of the field indicates whether the respective form exists.

Influencing the Output

The output of the table of contents can be controlled via parameters that are located in the Table of contents group box on the Graphics output tab. This tab is also offered in the Graphics output parameters dialog box.

EPLAN 5 Basic Training

Graphics output parameters

Tabs: Graphics forms | Graphics output

General

Number of blank pages: 0

Round up to page: 1

Terminal/interconnect diagrams

☐ Create subpages

Character for subpages: a = Alphabetic (lower case)

☐ Plot cable length

Symbol to mark conductors:

☐ Suppl. data in the 'cable type' field

☑ Break up macros

Terminal diagram and wiring list output:

☑ Common H-L assignment/loc. designation

Interconnect diagram output:

⦿ Common H-L assignment/loc. designation

◯ Higher-level assignment/location designation of the cable designation

◯ Higher-level assignment of the cable designation

Table of contents

☑ Create subpages

Character for subpages: 1 = Numerical

Number of blank pages: 0

Round up to page: 0

Output method: Entire output

[OK] [Cancel] [Apply]

EPLAN 5 Basic Training

The following options are available:

Create subpages:
This parameter is important if the table of contents is longer than one page. You use it to specify whether subsequent pages will be numbered as subpages or will be sequentially numbered.

Character for subpages:
This parameter becomes important if the Create subpages check box was activated. You can then specify the numbering style to be used for subpages.

If subpages have numerical designations, EPLAN can manage up to 9 subpages for each primary page. In this case, the table of contents can have a total of 10 pages for each primary page (the primary page itself + 9 subpages).

If the designations on subpages are alphabetical, EPLAN can manage up to 26 subpages for each primary page ("A" through "Z" or "a" through "z"). In this case, the table of contents can have a total of 27 pages for each primary page (the primary page itself 26 subpages).

If the project table of contents is too large to be displayed on the available subpages, EPLAN automatically creates a follow-on page. In this case, EPLAN recommends a starting page for the insertion of the table of contents, so that all pages in the table of contents appear sequentially in the project.

Number of blank pages:
Use this parameter to specify how many pages will be left blank between the most recently used page and the table of contents.

Round up to page:

This parameter, which operates together with the Number of blank pages parameter, is used to specify the page number from which you want the table of contents to start.

Both parameters are set as follows:
"Number of blank pages" = "4"
"Round up to page" = "10"
Case 1:
Let us assume the last page number is "47" --> The table of contents will then start on Page "61".
Explanation:
The last page number is "47". You have to add 4 blank pages, which results in "51" as last page number. This result is rounded up the next full "10" so that the last page is "60". Therefore, "61" is the first page number of the table of contents.
Case 2:
Let us assume the last page number is "46" --> The table of contents will then start on Page "51".
Explanation:
The last page number is "46". You have to add 4 blank pages, which results in "50" as last page number. This result is rounded up the next full "10" so that the last page number is "50". Therefore, "51" is the first page of the table of contents.

Output method:

Use this parameter to adjust the output of the table of contents to your project type. It is, for example, possible to output a separate table of contents for each higher-level assignment of your DIN project.

The dropdown list offers the following options:

- **Entire output**: This option enables you to output the entire table of contents at a defined position.
- **By higher-level assignment**: This option allows you to output the table of contents separately for each existing higher-level assignment.
- **By location designation**: This option allows you to output the table of contents separately for each existing location designation.
- **KKS:** Using this option, you can output the table of contents separately for each document and documentation type.
- **Entire output and by higher-level ass.**: This option enables you to output the entire table of contents at a defined position and, in addition, by higher-level assignments.
- **Entire output and by location des.**: Using this option you can output the entire table of contents at a defined position and, in addition, by location designations.
- **Entire output and KKS**: This option enables you to output the entire table of contents at a defined position and, in addition, KKS-oriented.
- **Manual**: Using this option you can highlight the pages to be output in the page overview of graphics editing and manually define the page designation, the starting page, and the output positions.
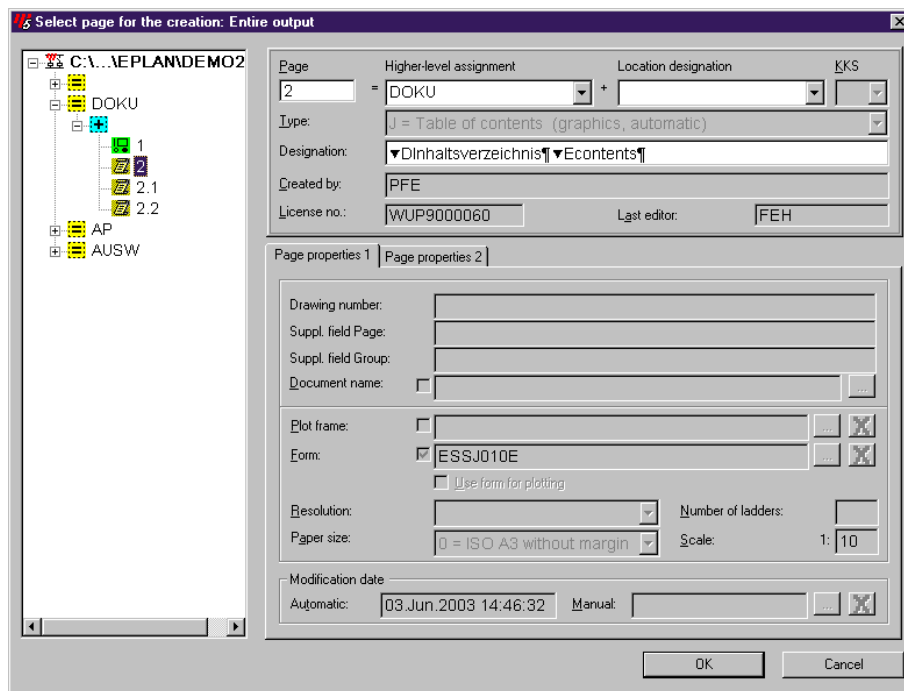
Creating and Deleting a Table of Contents

Project > Graphics editing, output > Extras > Table of contents > Create

The table of contents is an overview of all pages contained in the project. You create it by following the menu path **Extras > Table of contents > Create** in graphics editing.

EPLAN then reads the pages of your project and searches for a free space at which the directory can be inserted. You can either accept or change the page EPLAN suggests as the first page.

EPLAN 5 Basic Training

Check the results of the creation by calling the table of contents pages up in the graphics editing module. You can change the parameter settings at any time and repeat the generation run. The previous table of contents pages will be deleted permanently from the project.

Furthermore, you can remove the table of contents from the project by following the menu path **Extras > Table of contents > Delete**.

EPLAN 5 Basic Training

Updating a Table of Contents

If you had already created a table of contents and added additional pages to the project afterwards, the table of contents must be updated.

To this purpose, EPLAN provides an update function, which can be called up in graphics editing under **Extras > Table of contents > Update**.

This function is also used if the page designation or the name of the creator was changed on an already existing page.

EPLAN 5 Basic Training

Backing up Data

Much has been written about backing up your data. We will only spend a few minutes adding our thoughts. Losing data as a result of, for example, damage to the hard drive, a virus, or accidental deletion can have disastrous results for your company. In order to lose as little as possible if the worst happens, you should:

- Back up all projects that you are working on daily.
- Back up projects prior to initiating global processing functions such as numbering and evaluation runs.
- Back up completed projects.
- Back up master data such as symbol files after any modification.

Utilities > Backup

To access the data backup menu, first go to the main EPLAN menu, then select **Utilities** and **Backup**. Before describing the various data backup functions we will start by explaining a number of terms used when working with the data backup module.

Terminology

File group

Project backup tasks differentiate between project-specific and non-project-specific data (master data). File groups are stored behind the directories defined for EPLAN, and each group contains files associated with specific task steps such as the macros group, whose files are located behind the `\EPLAN4\M` directory.

ePLAN®5

EPLAN 5 Basic Training

Documentation Database

This database is managed by the EDOC program function and is used to provide an overview of the projects. In particular, this includes information about when and where a project was backed up, filed-off, or archived. For detailed information on EDOC, please refer to the section "Editing the Project Documentation" earlier in this chapter.

In the subsequent section you will find information on the various data backup functions.

- Defining Default Settings
- The Backup Dialog Box
- Backing up and Reloading Data
- Compressing and Extracting Projects
- Executing Special Functions of the Data Backup.

Defining Default Settings

Utilities > Backup > Parameters

Prior to a data backup, you may have to make various default settings. For this purpose, the Data backup utility offers the following menu items or parameters:
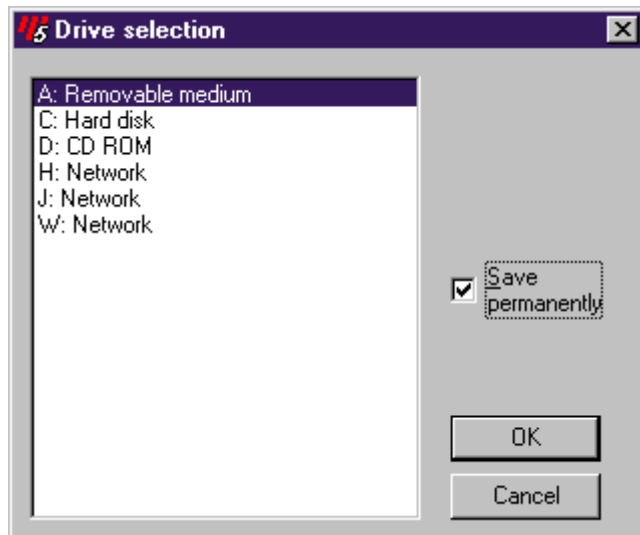
- Drive
- Input / Output.

If you want to carry out a data backup via a control file, you must adjust the control file beforehand. The following section gives details on the above-mentioned parameters and the control file for the data backup.

EPLAN 5 Basic Training

Drive:

You can use the **Drive** menu item to temporarily change the "Data exchange drive" (see below) specified in the input / output parameters.

To do this, select the drive on which you want to carry out a data backup or from which you want to reload data in the Drive selection dialog box. All the available drives are displayed.
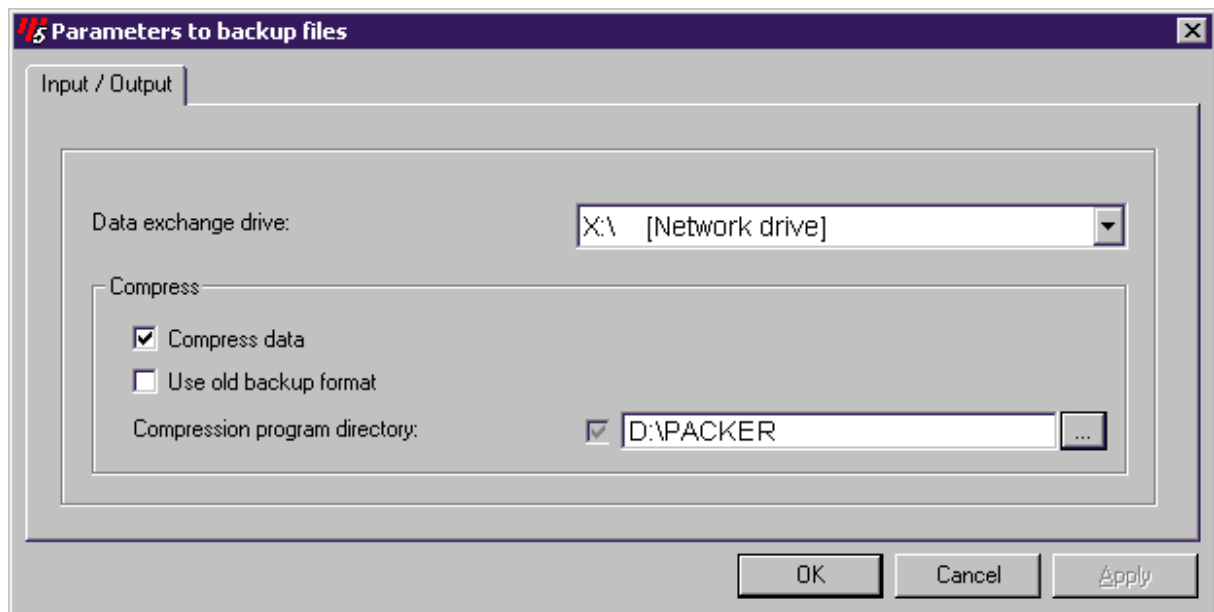
If you click on the Save permanently check box, the drive selected here is applied as Data exchange drive in the input and output parameters.

Input / Output:

Use the **Input / Output** menu item to define the workstation-specific parameters for the data backup. The following parameters are available:

- Data exchange drive
  (This is the drive on which the data are saved during the data backup process or from which the data are read during a reload process.)
- Compress data
- Use old backup format
- Compression program directory.

For further information on these parameters refer to the section "Specifying I/O Parameters" in this User Guide.

Editing the Control File DATSICH.INF

In order to specify the extent of the backup you can the edit the control file DATSICH.INF (in the customer-specific directory for the master data). Use the menu items **Extras > Edit control file** in the **Data backup** dialog box. This function allows you to define the extent of backup separately for each data group.

Utilities > Backup > Extras > Edit control file

EPLAN 5 Basic Training

The control file is an ASCII file that must be built up on a line-by-line basis. Each line contains either a file specification or a control sequence.

All **control sequences** must begin with a period and define the respective file group. The respective file group character is entered after the period. The following assignments apply:

P   Projects
N   Master data
M   Macros
L   Parts data
F   Print and control forms
O   Personal and workstation parameters.

Each control sequence line is followed by one or more lines containing **file specifications**. The use of the wildcard characters "*" and "?" is allowed in accordance with standard DOS conventions. "*.*" means that all the files of the directory are to be backed up.

Example of a `DATSICH.INF` file

```
.P              Project directory
   seiten.*         Graphics database
   projekt.ppd      Project parameter file
   ort.*            DIN file of location designations
   anlage.*         DIN file of higher-level assignments
   .N               Directory for master data
   *.*                  all files
   .M               Macro directory
   ESS????D.MSF     Page macros graphics pages
   .L               Parts directory
   *.*                  all files
   .F               Directory for print and control forms
   EPLAN\*.*            All EPLAN form files
   EPPS\*.*         All ERP form files
   .O               Personal and workstation directories
   *.*                  all files.
```

EPLAN 5 Basic Training

Files specified with a preceding ":" will **not** be backed up. All other listed files will be backed up.

If a colon is used for the identification, only the combinations of a colon with the "`file_name.*`" or with the "`*.file_extension`" are allowed.

💡 Examples of the identification with a colon for an extract from another `DATSICH.INF` file.

```
.P              Project directory
  :backup.*        e.g. backup.spl
  :*.err           e.g. Epl0004.err
  *.*
  .M              Macro directory
  "..."
```

This means that, in the project directory, all files except for the backup files and the error files are backed up.

The Backup Dialog Box

The Backup dialog is called up when you have selected a project and wish to save it or to send it by e-mail or when you wish to save non-project-specific data such as macros.

The various options of this dialog box will be explained below. Depending on the backup selected you do not always have all options available.

EPLAN 5 Basic Training

Specifying the Backup Method

There are three backup methods available:

Backup:
Backing up copies the project to the target data medium. In other words, the project remains available in exactly the same form.

File off:
The file-off function is used to copy the project to the target data medium for additional external editing or modification. The original project is then locked to prevent any modifications being made to it. Thus, only the filed-off project can be changed. You can, however, open the original project on a read only basis.

Filed-off projects are marked by special graphics symbols in the project selection windows. If the project has been compressed additionally, a different symbol is used. (Cf. the listing of symbols in the section "Backing up and Reloading Data" in this chapter.)

Archive:
Archiving is used to file completed projects to a target data medium. The entire original project is deleted from the current hard disk, leaving only an information file. Archived projects are also identified by a special symbol in the project selection windows.

Specifying the Extent of the Backup

You can specify the extent of the backup operation. That is, you can define which files are to be included during data backup. Your options here are:
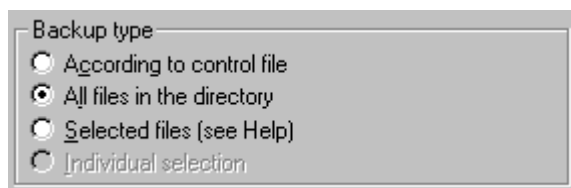
EPLAN 5 Basic Training

According to control file:
When this option is used, the system backs up only those files in the directories specified in the control file. The control file allows you to precisely define which files from which file groups are to be backed up.

The section on "Editing the Control file" in the previous chapter provides you with more detailed information about the control file structure.

All files:
This option backs up all files belonging to the respective file group. This option is primarily used when archiving data.

Selected files:
If you use this option, you can only back up those files that are required for project pages. Files derived from the project pages, such as cross-references, terminal diagram, and bill of materials, are not included in the backup.

Individual selection:
This radio button can only be activated if master data, macros, parts data, print and control forms, or data from the personal directory are backed up. You can then individually select the directories and files which are to be included in backing up.

Deleting files no longer required
The following settings are possible here:
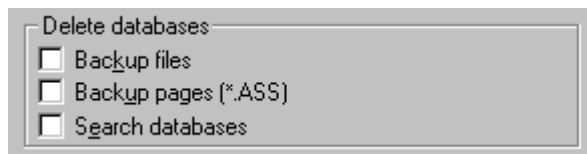
Backup files:
This includes various backup files that are created during the execution of the program, such as the backup pages, which are created before a compression is carried out, etc.

Back-up pages (`*.ASS`):
Select this check box if you want to delete the backup pages that EPLAN generates automatically in accordance with the setting of the personal 'Number of page in page buffer' backup parameter.
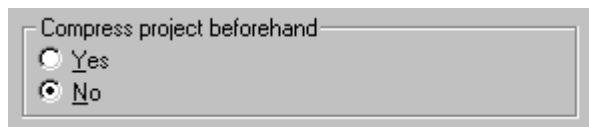
EPLAN 5 Basic Training

Search databases:

This check box can be used to specify that the project-specific search results which are written into the search database during certain actions are to be deleted.

Compressing a Project

You can use the radio buttons in this group box to compress your project during the data backup.

```
┌─ Compress project beforehand ──────────────┐
│   ○ Yes                                     │
│   ◉ No                                      │
└─────────────────────────────────────────────┘
```

This compression is a completely separate action from the project compression function, and is performed by a separate EPLAN module which reorganizes the project database. The function irretrievably removes any deleted pages from the project and removes gaps in the database. The project can be examined at the same time for data consistency.
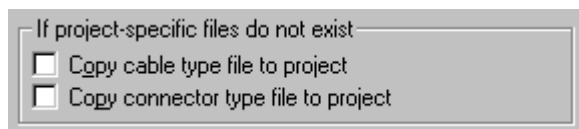
In addition, you can continue to edit a compressed project with the standard EPLAN graphics functions. Only the deleted pages can no longer be accessed.

☞ The database compression can be started in the EPLAN main menu by selecting **Project > Compress page database.** The section on "Revising the Page Database" in this documentation will provide you with further information on this topic.

Backing up General Type Files

If you have used general, not project-specific, cable- and connector-type files in your project, you can use the two check boxes in this area to specify whether these are to be backed up with the project data.

```
┌─ If project-specific files do not exist ──────┐
│   ☐ Copy cable type file to project          │
│   ☐ Copy connector type file to project      │
└───────────────────────────────────────────────┘
```

EPLAN 5 Basic Training

Backing up Associated Files

When you back up projects, you can also include certain files from other file groups in the project. These files are required to display a project. These include symbol files, combined and variable drawing macros and forms for terminal diagrams, interconnect diagrams, etc. The following possibilities are available:
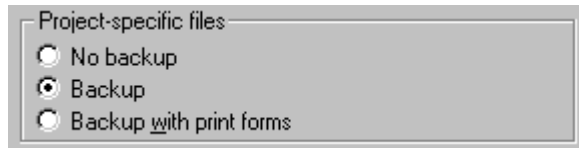
No backup:
The associated files will not be backed up. Select this option if you have already separately backed up the other file groups.

Backup:
The associated files from the N: Master data and M: Macros file groups are backed up. Their names are read from the project parameters (symbol files, plot forms) and the project pages (symbol macros). Select this option if you want to send a project to another EPLAN user, e.g. to allow him to edit it.

Backup with print forms:
This option will include all the print forms used in the project (as defined in the parameters), together with the master data and the project macros.

Updating the Documentation Database

In order to update the information in the documentation database after a data backup, you must create a log and activate the log function as well as create an EDOC database before the projects are backed up and reloaded.

Backing up data also has an effect on the documentation database, since, for example, archiving projects must be recorded in this database. The database contains input fields for projects in which specific information concerning data backup is recorded. If, in addition, you have also activated the logging function, the data backup procedures will also be recorded in the log.

ePLAN 5

EPLAN 5 Basic Training

The fields in the right-hand area of the Backup dialog box have the following meaning:

Backup number:
You can document up to three backup processes for each project. Each backup action is given a number. You can enter this number in the input window that appears when you initiate a data backup. The entry associated with the number you input is replaced in the documentation database.

Backup medium:
The backup medium on which backing up is to be carried out is entered in this field. The possible entries can be selected from the dropdown list box.

Number of media:
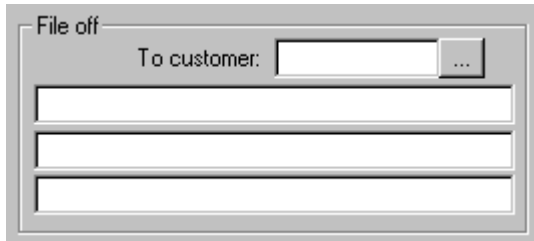Use this field to enter the number of media needed.

Stored at:
You can enter the storage location of the medium into this field, e.g. Project archive, Data safe, Personal archive. In addition, the data and time at which the backup was carried out are entered in the documentation database.

Filing off Projects

If you have activated the File off for external processing radio button under Backup method, you can enter the corresponding information for the project in the File off group box.

Use the **[...]** button to launch the **Customer selection** dialog in which you can select the customers for the To customer field. In this dialog box all customers are listed who are stored in the customer master data of the parts management.

Saving the Defaults of the Backup Settings

In order to save the current backup settings as the default values, click the **[Save default setting]** command button as soon as all the check boxes and radio buttons have been set in accordance with your requirements. Whenever you call up the **Backup** option again, these settings are then offered as the defaults.

Selecting a drive or sending an e-mail

Using the dropdown list box of the To field you can select the drive on which the data are to be saved. If the data are to be sent as e-mail the @E-mail entry must be selected from the list. How to send e-mails is explained under "Sending projects by e-mail" in the section "Editing Projects" in this chapter.

EPLAN 5 Basic Training

Backing up and Reloading Data

Backing up a Project

When backing up projects to diskettes, we recommend that you have an adequate number of already formatted diskettes available before you begin the backup procedure. Then select the **Backup > Projects** menu items in the Data backup module.

Irrespective of the project you are currently editing it is possible to select any project on the current drive or via the **[Drives]** button also on another drive in the subsequent Selection of the projects to be backed up dialog. Use **[Reset]** to set the selection back to the project that was originally set in the Project selection dialog box.

Graphics Symbols in the Project Selection Function

The following graphics symbols are used to **display the backup mode** of a project in the tree view of the selection dialog box:

Symbol	Backup mode
	Current projects
	Compressed projects
	Archived projects
	Filed-off projects
	Filed-off projects, which have been compressed.

Afterwards the Backup dialog is called up in which you can specify the various backup parameters. Also refer to the information on the Backup dialog box. Confirm with **[OK]** to start the backup action. If necessary, you will be asked beforehand to insert a backup diskette.

If the system notices that the backup medium is not empty, a message is displayed and you can decide whether you still want to use it for the backup.

The result of the data backup is documented in the message management and output to a window. When you confirm with **[OK]** you will be returned to the backup module.

EPLAN 5 Basic Training

If all your project files do not fit on a single diskette, the additional diskettes will be assigned the designation `P` and be given a sequential number. Note the name on the label of the respective diskette.

Copying Projects for Data Exchange

When you copy projects to diskettes for the purposes of exchanging data, you should take note of the following points when backing up the project:

- If you use your own, unique symbol files, macros, plot forms, etc. in your project, you must also supply the corresponding files so that the recipient can actually work with the project. To this purpose select the Backup or Backup with print forms radio button in the Project-specific files group box of the Backup dialog box.

- If you only use those files that form part of the standard EPLAN delivery to create your projects, you do not need to select the options mentioned above for backing up associated files.

Backing up Further Data

In addition to those data already mentioned, the following file groups can also be backed up with the data backup module (in each case, the base directory is shown):

Master data	`\EPLAN4\N`
Macros	`\EPLAN4\M`
Parts data	`\EPLAN4\L`
Print and control forms	`\EPLAN4\F`
Personal and workstation data	`\EPLAN4\O`

Once the desired menu item (e.g. Macros) has been selected from the **Backup** menu, the Backup dialog box appears. Compared to the project data backup only a few parameters can be specified here.

In the subsequent Directory selection dialog you can select the directory whose files are to be stored. If the Individual selection option was set in the Backup type group box of the Backup dialog box, a file selection must be made afterwards.

The further procedure to back up data corresponds to the procedure when backing up project data.

EPLAN 5 Basic Training

Reloading Projects

To reload data, use the **Reload** menu of the Data backup module.

 Utilities > Backup > Reload > Projects

If necessary, you will be asked to insert a diskette. After project selection in a subsequent dialog box you can decide while reloading projects whether the project is to be reloaded under its original name or whether you want to assign a new name.
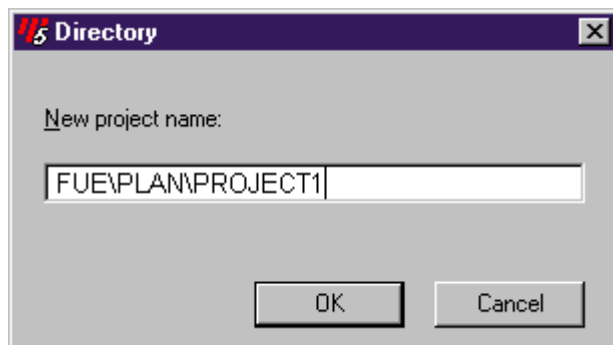
If you confirm the original name, the project files are written back to the original directory. If you enter a project name that already exists (or confirm of the original name), a window appears asking you if you want the project directory to be overwritten.

When reloading projects for which "project-associated files" were backed up as well you will we prompted in subsequent dialog boxes how to proceed with the associated files.

The reloading result is documented in the message management and output to a window. Click **[OK]** to return to the data backup module.

EPLAN 5 Basic Training

Specifying Directories for Shared Data

When reloading projects, an additional dialog box is called up once the project name has been specified. The Specify directories dialog box only appears for projects for which the "project-specific files" were included in the backup. Enter the directories manually here, or use the **[...]** buttons that will open a selection dialog box.
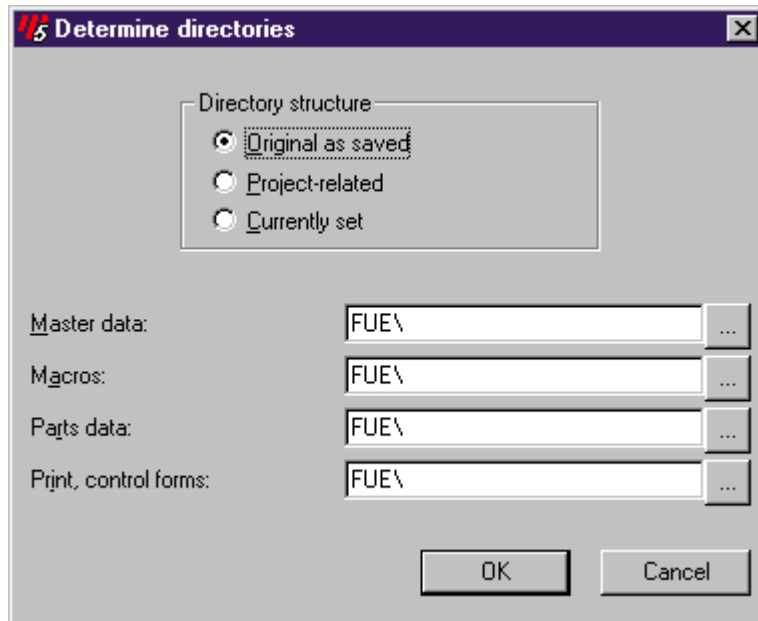
Use this dialog box to specify the directories for the shared data. By doing this, you also determine the directories into which the associated files can be copied afterwards.

Select an option from the Directory structure group box. The following options are available:

Original as saved:
The directory structure of the reloaded project is applied.

Project-related:
The directory structure is copied from the directory that was specified in the previous dialog box for the project name.

EPLAN 5 Basic Training

Currently set:
The directory structure of the currently selected project is applied.

The directories of the selected directory structure are shown in the fields below. These directories can be changed which is useful if a different directory is to be used e.g. for the macros. When you click **[OK]**, the reload procedure will be continued and you will be asked what should be done with the associated files.

Reloading Additional Data

Reloading other data (master data, macros, parts data, print and control forms, personal and workstation files) is done in the same way.

After having selected the corresponding menu item from the **Reload** menu, you must choose the directory of the data to be reloaded in the selection dialog box. Afterwards you may change the target directory, if required. When doing so, the respective base directory e.g. \EPLAN4\M for macros cannot be changed. When finished, click on **[OK]** to return to the data backup module.

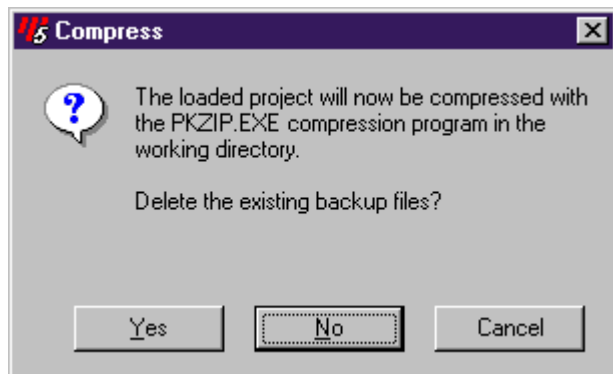# Compressing and Extracting Projects

Compressing Projects

You can reduce the space requirements of project files by compressing projects. This function uses a so-called "compression program" to compress the files. Decompressing the files extracts them from the compressed file.

☞ Please note that you cannot edit compressed projects!

Select the **Backup > Compress** menu items to compress the current project in the current working directory. To do this, the Clean project dialog box is launched. In this dialog you can specify which of the files that are not needed for the data backup can be deleted. After the prompt whether the existing backup files are to be deleted the compression process is started.

ePLAN®5

☞ In project selection, compressed projects are identified by a special graphics symbol (🖳) as well as by the text "Compressed" in the project properties.

💡 Example for the display of a compressed project in the project selection
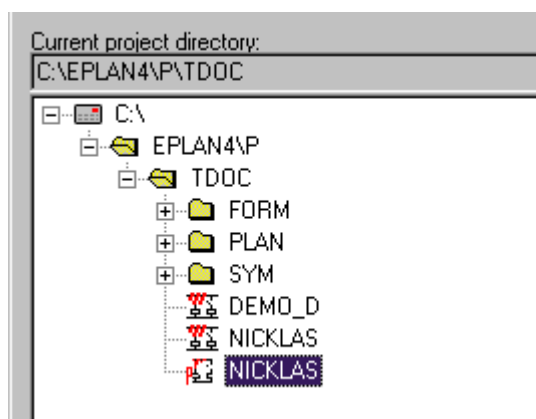
Extracting Projects

If you select a compressed project for graphics editing, EPLAN points this out and offers to extract it in a subsequent dialog box. As an alternative, the decompression can also be directly started via the **Reload > Decompress** menu items of the Data backup module.

![ePLAN 5]

EPLAN 5 Basic Training

Please note that in case you decompress data that were packed by the ARJ compression program in an earlier EPLAN version (prior to EPLAN 5.50), you must set a certain workstation-specific parameter. If this is the case, specify the directory in which the ARJ compression program is loaded in the input and output parameters. As an alternative, the directory can also be determined by a corresponding entry in the PATH system variable (Control Panel).

👉 Notes:
Projects can also be compressed and extracted within the project management. To do this, select the **Select, create, copy project** option under the **Project** menu item to call up project management. Highlight the desired project(s) in the directory tree, and select the respective **Compress** or **Decompress** menu item from the pop-up menu. Unlike the process in the data backup module the Compress or Decompress queries will not appear but the projects will be compressed or extracted immediately.
If the project-specific directories do not correspond to the current directories for the compression, the Directory selection dialog box comes up. Select the Project-related setting here to backup the correct directories during the compression.

New ▶
Rename
Copy
Delete
Edit
Save
Update
Drives
Selection ▶
Compress
Send by e-mail
Show user

EPLAN 5 Basic Training

Executing Special Functions of the Data Backup

Reloading the Backup

The Reload backup dialog shows a list of the last backup copies of the entire project. The display contains the user, date, time, and the executed function.

To retrieve a backup copy highlight the corresponding line in the dialog box. Then click on the **[OK]** command button. The process is started and the dialog box closed. If you wish to delete a copy click on the **[Delete]** command button after having highlighted the corresponding line.

The Reload backup dialog box can be called up as follows:

- In the data backup module via the menu path **Extras > Backup management**
- In the cross-reference module and in the terminal and interconnect diagram module by following the same menu path
- In the Move, Number, and Position functions of the generation runs by using the **[Undo]** button.

In order to undo changes it is possible to create a backup copy for the project before executing the Correction function, Move, Number and Position functions. To do this, the Create backup check box must be activated before carrying out the respective function. A backup is also generated when the page database is compressed.
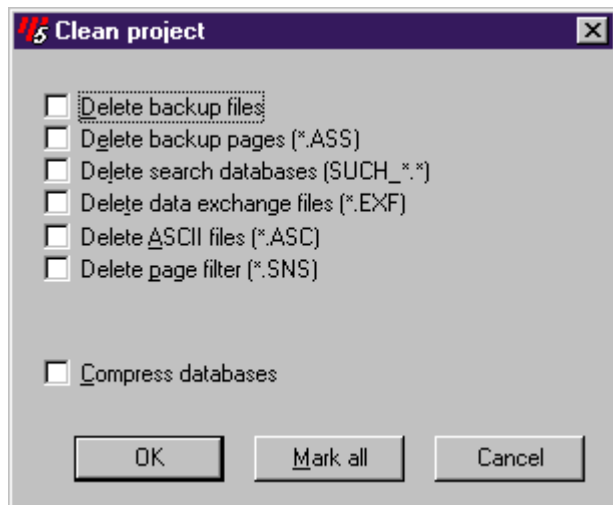
☞ Notes:
In EPLAN COMPACT, the backup management is not included in the backup module, but can only be launched by clicking the **[Undo]** button of the generation runs.
You can specify the number of backup copies to be created in the parameter management function under **Personal > Backup parameters > Number of backups**.

Cleaning up Projects

Select **Extras > Clean project** to activate the Clean project dialog box used for project maintenance. Files that are not needed for the data backup can be removed from the current project through this dialog.

ePLAN®5

EPLAN 5 Basic Training

Clean project

☐ Delete backup files
☐ Delete backup pages (*.ASS)
☐ Delete search databases (SUCH_*.*)
☐ Delete data exchange files (*.EXF)
☐ Delete ASCII files (*.ASC)
☐ Delete page filter (*.SNS)

☐ Compress databases

[OK]   [Mark all]   [Cancel]

Furthermore this dialog box contains a function for compressing the databases. Using this function the project requires less storage space and is examined at the same time for data consistency. Database compression was already discussed under "Compressing a project" in the section "The Backup Dialog Box".

The **backup files**, **backup pages,** as well as the **search databases** are also described in the above-mentioned section since the deletion of these files can also be carried out in the Backup dialog box (see subsection on "Deleting files on longer required").
The following additional files can be removed through the Clean project dialog box:

Data exchange files (`*.EXF`):
When exporting a project to the data exchange format EXF the entire project is converted to a file. This file that, by default, is called `EPL_ASC.EXT` and is stored in the project directory can therefore become quite large and should be removed before the data backup is carried out.

ASCII files (`*.ASC`):
Through the corresponding check box, ASCII files that may have been created in different ways are removed from the project directory.

Page filter (`*.SNS`):
A page filter file is created in the project directory if a page selection is permanently saved in graphics editing.

To clean the project activate the corresponding check boxes and confirm with **[OK]**. If all functions are to be carried out click the **[Mark all]** button first and then click **[OK]**.